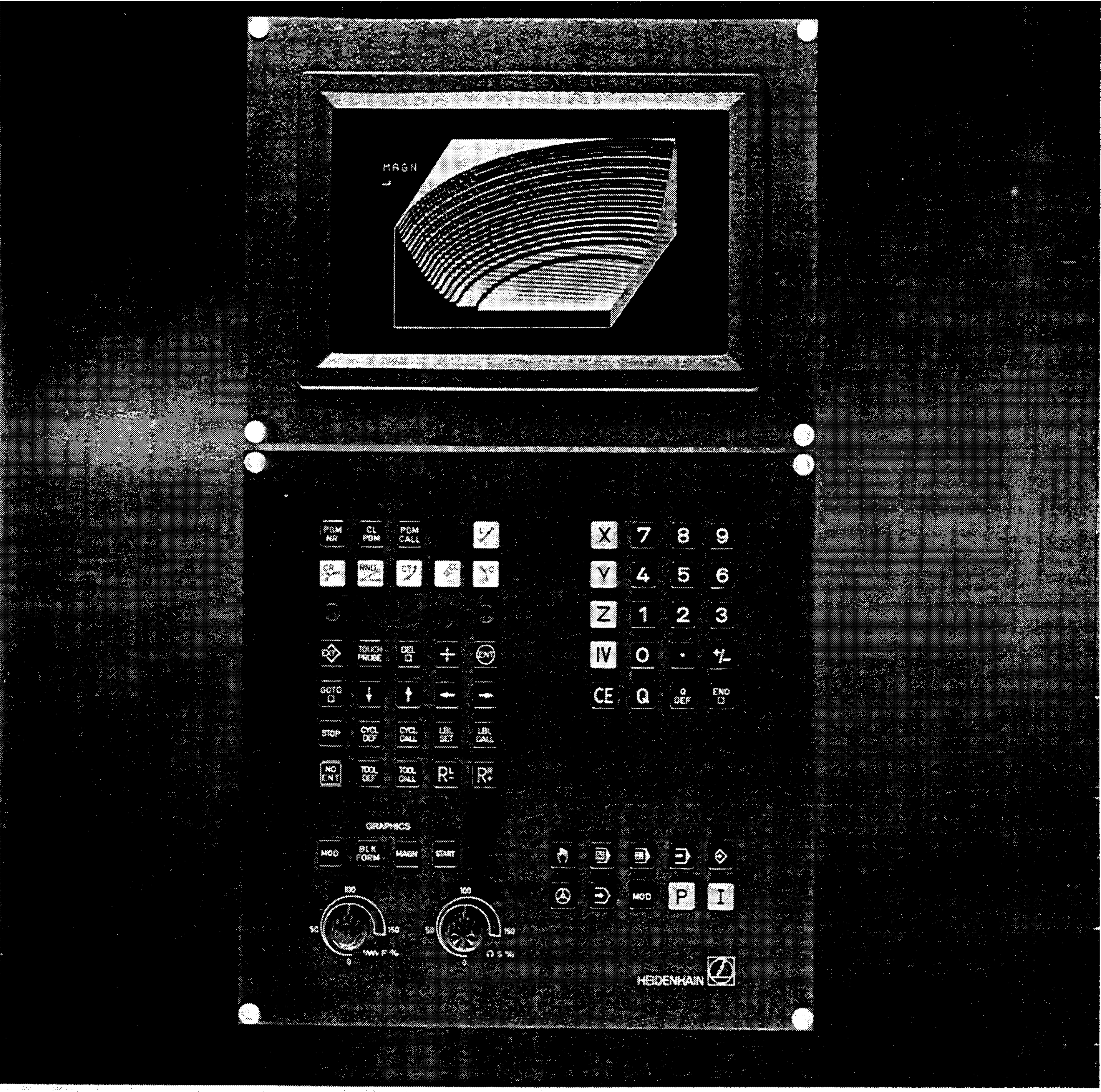


HEIDENHAIN

User's Manual
HEIDENHAIN-Conversational Programming

TNC 306

August 1993

Screen displays

```
PROGRAM RUN /FULL SEQU. 100

38   LBL   2
_______________________________________________
39          Z+Q20
                          R0 F100     M36
_______________________________________________
40          Z+2
                          R   F       M37
NR LV HV GV T-ON TF SV AJD  ET
25 99 9  99 100  10 99 1,5  5
----------- 00:02:31 -----------
NOML. XN      +0,000  YN      +0,000
+     Z       +1,347  C       +0,000

300/25-1/25          WTG      11,347
                     ROT +    20,000

T9999 Z UM 0,100     F   114    M37
```

Operating mode
Error messages

Preceding block

Current block

Next block

Eroding parameters

Machining time

Status display

Status display:

ACTL.:	Type of position display, switchable with MOD (further displays: NOML, DIST., LAG – see chapter "General Information")
X . . .	Position coordinates
Y . . .	
Z . . .	
C . . .	
✱:	"control is started" display
N:	Datum shift, shown as an index on the shifted axis.
S:	Mirror image, shown as an index on the mirrored axis.
ROT:	Basic rotation of the coordinate system
SCL:	Scaling
CC:	Circle center or pole
WTG:	"Way To Go", distance remaining to be eroded
T . . .:	Called tool
Z:	Tool axis
UM:	Tool undersize
F:	Feed rate
M:	Miscellaneous function (M03, M04, M05, M13, M14)

Guideline for procedure from preliminary operations to workpiece machining

Sequence	Action	Operating mode	Cross reference	Page
1	Select electrode	–	Workpiece drawing	–
2	Set datum for workpiece machining	–	Workpiece coordinates	A19
3	Switch on machine	–	Machine operating manual	–
4	Traverse reference points (homing the machine)	–	Switch on	M1
5	Clamp workpiece	–	Clamping instructions	–
6	With electrode: datum setting and compensation of workpiece misalignment	Manual	Datum setting with probe functions	M4
7	Enter program – by keying in or from external storage device	Programming and editing	Back fold-out page, program example; Programming and editing	P1
8	Test program (without axis movements)		Test run	P112
9	Graphic program simulation (without axis movements)	Program run	Test graphics	P113
10	Test run without electrode in single block mode	Program run, Single block	Program run	M21
11	Optimize program if necessary	Programming and editing	Editing functions	P3
12	Insert electrode and machine workpiece automatic program run	Program run, Full sequence	Program run	M21

Operating Panel TNC 306

Machine Operating Modes

	Manual operation
	Electronic handwheel
	Positioning with manual data input
	Program run, Single block (test graphics)
	Program run, Full sequence (test graphics)

Programming Modes

	Programming and editing
	Test run with graphics

Program Management

PGM NR	Name/select a program
CL PGM	Clear program
PGM CALL	Programmable program call
EXT	External program input and output
MOD	Supplementary operating modes

Test Graphics

MOD	Graphic display modes
BLK FORM	Define blank form, reset blank form
MAGN	Magnify detail
START	Start graphic simulation

Override

S %	C axis rpm override (with M03, M04)
F %	Feed rate override (during positioning and M37)

Programming

Entering the Workpiece Contour

L	Straight line
CC, C	Circle with known center
CR	Circle with known radius
CT	Circle with tangential transition
RND	Round corners/ Tangential contour approach and departure
TOOL DEF, TOOL CALL	Define/Call an electrode
RL, RR	Specify mode of electrode compensation
CYCL DEF, CYCL CALL	Define/Call a cycle
LBL SET, LBL CALL	Label/Call a subprogram and program section repeats
STOP	Programmed stop/Terminate program
TOUCH PROBE	Probe

Entering and Editing Values

X . . . IV	Axis keys
0 . . . 9	Number keys
. , +/-	Decimal point, sign change
P	Key for polar coordinates
I	Key for incremental dimensions
Q, Q DEF	Enter parameter instead of a number, Define parameter
	Transfer actual position to memory
GOTO	Cursor keys, Jump to a certain block or cycle
NO ENT, ENT, END	No entry, Enter data, Terminate block entry
CE	Clear entry
DEL	Delete block

Contents

This User's Manual describes software versions 260 03x 04 and 260 05x 04.

General Information (A)

Introduction

Description

The TNC 306 from HEIDENHAIN is a shop-floor programmable contouring control with three or four axes for ram-type electrical discharge machines. It is conceived for the "man at the machine," featuring conversational programming and excellent graphic simulation of workpiece machining. Its background programming feature permits a new program to be created (or a program located in the memory to be edited) while another program is being executed. Besides fixed cycles, coordinate transformations and parametric programming, the control also includes path functions for spark erosion and edge-finder functions for "electronic" workpiece alignment with the electrode.

Files (part programs, erosion tables, etc.) can be output to peripheral devices and read into the control via the RS-232-C data interface, allowing programs to be created and stored externally.

Compatibility

This control can execute programs from other HEIDENHAIN controls, provided they contain only the functions described in this manual.

Structure of manual

This manual addresses the skilled machine operator and requires appropriate knowledge of non-NC-controlled die-sinking electrical discharge machining.

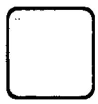
TNC beginners are advised to work through this manual and the examples systematically. If you have already worked with a HEIDENHAIN TNC, you can skip familiar topics.

The sequence of chapters in this operating manual is according to control operating modes and key functions, as well as according to the logical working order:

- **Machine operating modes:**
  Switch-on – "electronic" alignment – set display value – machine workpiece.
- **Programming modes:**
  Enter program – test program.

Symbols for keys

The following symbols are used in this manual:

Empty square:  ... keys for numerical input on the TNC operating panel

Square with symbol, e.g.  ... other keys on the TNC operating panel

Circle with symbol, e.g.  ... buttons on the machine operating panel

The pages of this manual are distinctly marked with the relevant key symbols.

Typeface for screen displays

Program blocks and TNC screen dialogs are printed in this SPECIAL TYPE.

Introduction

Program Examples

The example programs in this manual are based on a uniform blank size and can be displayed on the screen by adding the following blank definition (see index "Programming Modes", Program Selection):

BLK FORM 0.1 Z X+0 Y+0 Z–40
BLK FORM 0.2 X+100 Y+100 Z+0

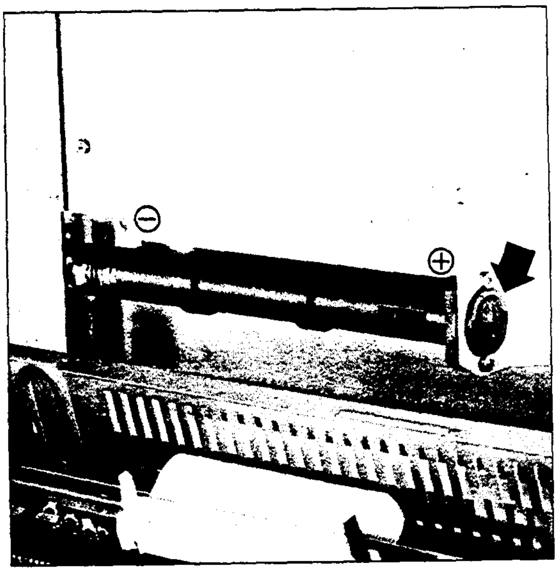
The examples can be executed on EDM with tool axis Z and machining plane XY. If your machine uses a different axis as the tool axis, this axis must be programmed instead of Z and likewise the corresponding axes for the machining plane.

Beware of collisions when executing the example programs!

Buffer batteries

Buffer batteries for the control

Programs and machine-specific data (machine parameters) are stored non-volatile via buffer batteries.



If the message
EXCHANGE BUFFER BATTERY
is displayed, new batteries are to be inserted.

Batteries should be exchanged each year.

Battery type:
Three AA-size batteries, leak-proof
IEC designation "LR6".

Battery exchange

To exchange the batteries the supply voltage can be disconnected.
The batteries are located behind the twist-lock cover in the power supply unit of the LE unit. To change the batteries the LE unit can be opened by the two snap locks.

Do not allow the swing frame to drop!

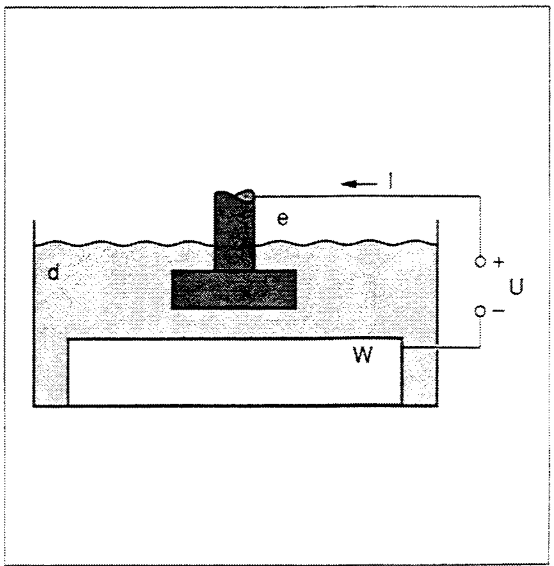
Fundamentals of Spark Erosion

The erosion process

Spark erosion is a thermal method of metal removal. Instead of removing material by mechanical means such as milling, an electrical discharge machine (EDM) melts and vaporizes the workpiece material.

The erosion process taken place in an electrically insulating fluid, the dielectric **d.** The workpiece **w** and the tool **e** are immersed in the dielectric and function as two electrodes placed under the voltage **u**.

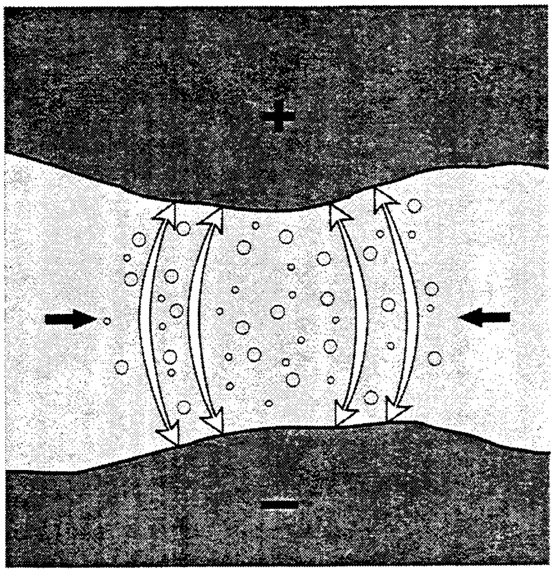
If the two electrodes are brought close enough together to cause an electrical current **I**, this current will take the form of sparks.



Discharge

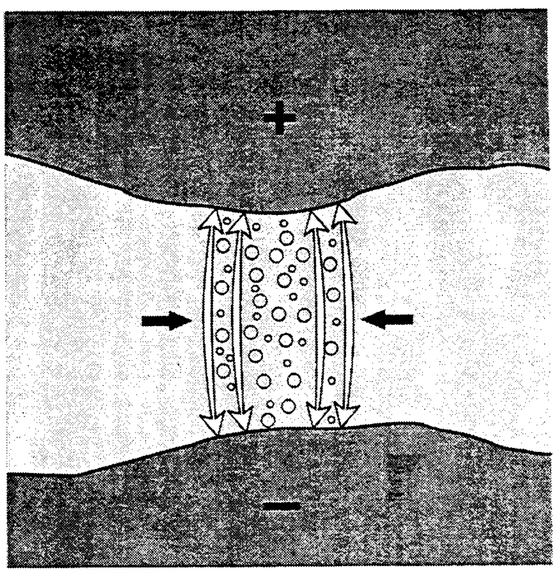
1. Applying an electrical voltage

If an electrical voltage is applied to the two electrodes, an electrical field arises that is greatest wherever the gap between them is the smallest. The voltage therefore concentrates all electrically conducting particles at this point.



2. Bridge formation

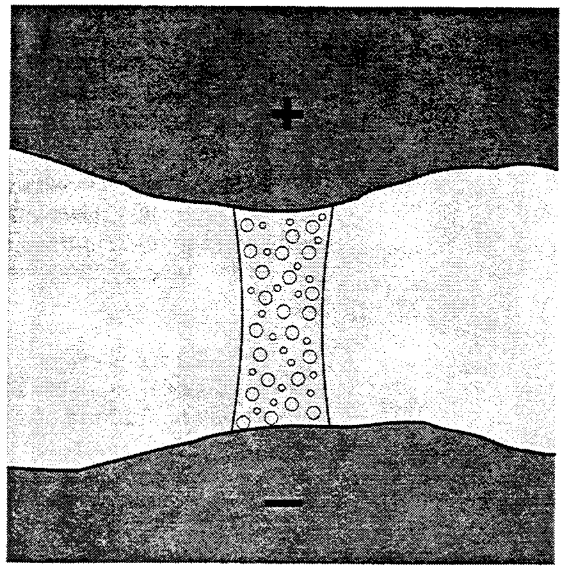
A type of bridge arises from the concentration and orientation of the particles in the direction of the electrical field.



Fundamentals of Spark Erosion

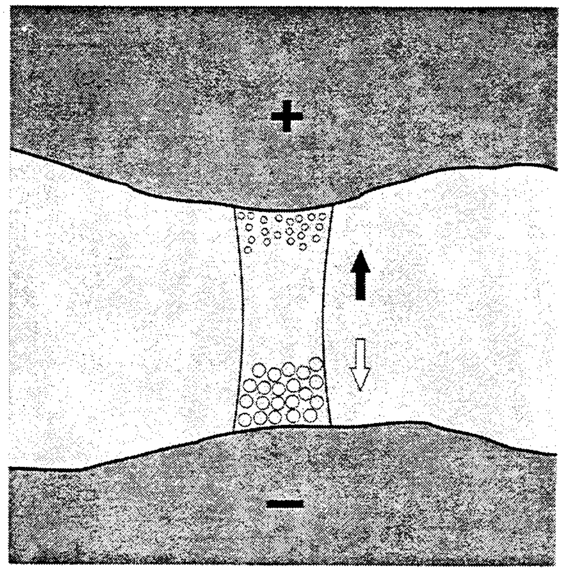
3. Discharge channel

After a certain delay (ignition delay time), a discharge channel forms over the bridge of particles.



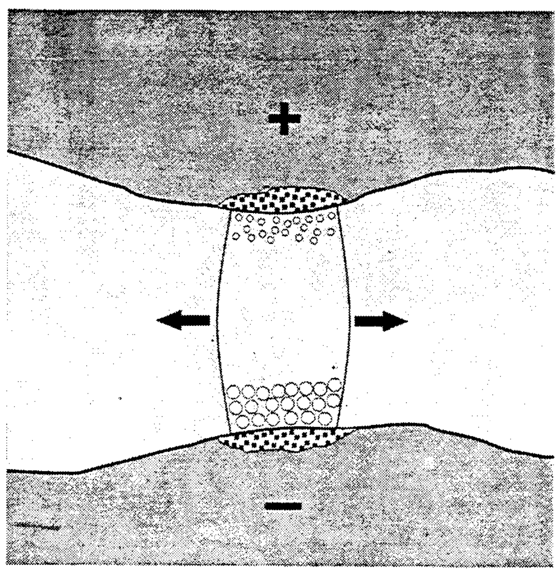
4. Electrical current

The particles begin to flow to the positive and negative electrodes. This flow of particles is an electrical current. The very high pressure and temperature arising from this current vaporizes the dielectric in the discharge channel.



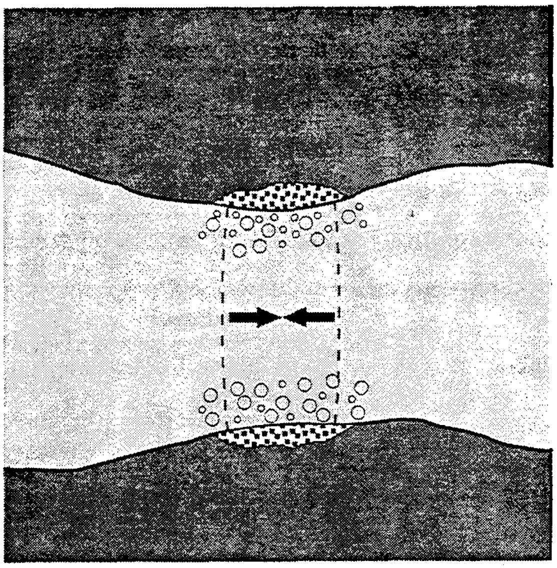
5. The discharge channel expands

The surfaces of both electrodes melt. The discharge channel expands, which decreases pressure and temperature.



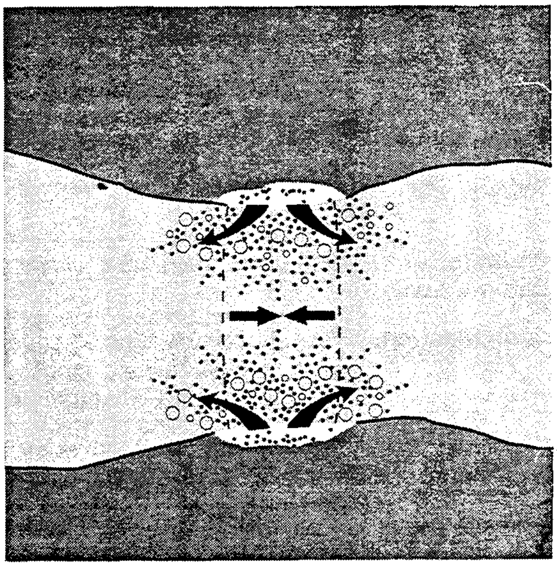
6. Current is switched off, discharge channel collapses

Switching off the voltage stops the electrical current and the flow of charged particles. The discharge channel collapses.



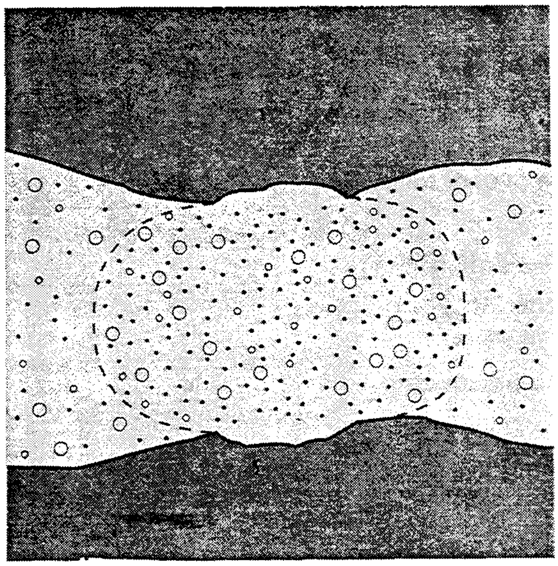
7. Molten metal is displaced and vaporized

When the discharge channel collapses, the implosion hurls the molten material away from the surface and into the dielectric, where it vaporizes.



8. Electrode particles in the dielectric

After discharge, particles from the electrodes remain suspended in the dielectric. A small crater remains on the surface of each electrode.



TNC 306
Brief description

Control type	Contouring control for 4 axes with erosion gap control
Traversing possibilities	Straight lines in 3 axes Circles in 2 axes Helix Helix with linear interpolation of the C axis (not on export version TNC 306 E)
Parallel operation	Programming and program execution simultaneously
Graphics	Graphic simulation in the "Program run" or "Test run" operating modes
Program input	In HEIDENHAIN format
Input resolution	Max. 0.001 mm or 0.0001 inch or 0.001°
Program memory	For 32 files (NC programs, erosion parameter tables and a datum table): approx. 88K byte RAM (battery-buffered)
Tools	Up to 254 tool definitions in a program: via TOOL DEF or tool definition cycle with tool compensation in max. 4 axes

Programmable functions

Contour	Straight line, chamfer Circle (input: center and end point of the arc or radius and end point of the arc), circle connected tangentially to the contour (input: arc end point) Corner rounding (input: radius) Tangential approach and departure from a contour
Program jumps	Subprograms, program section repeats, call of other programs
Eroding cycles	Generator definition, disk cycle for circular and square machining, orbital sinking, erosion with time limit, tool definition
Coordinate transformations	Move and rotate the coordinate system, mirror image, scaling
Probing functions	For electronically aligning and measuring the workpiece with the electrode in the "Manual operation" and "Electronic handwheel" modes of operation, as well as for programmed probing in the "Program run/ single block" and "Program run/full sequence" modes of operation
Parametric programming	Mathematical functions (= / + / − / x / ÷ / sin / cos / angle α from axis sections / $\sqrt{a}$ / $\sqrt{a^2 + b^2}$); parameter comparison (= / ≠ / > / <); PLC error; print; indexed data assignment
Traversing range	Max. ± 30 000 mm or 1181 inches
Traversing speeds	**Traversing speed: max. 30 m/min or 1180 inches/min** **C axis: max. 83 rpm**

Hardware

Component units	Logic unit, control panel and monochrome screen
Block processing time	1000 blocks/min (60 ms)
Control loop cycle time	4 ms
Data interface	RS-232-C/V.24 Data transfer speed: max. 38 400 baud
Ambient temperature	Operation: 0° C to 45° C (32° F to 113° F) Storage: −30° C to 70° C (−22° F to 158° F)

Error Messages

The TNC checks the inputs and status of the control and the machine.

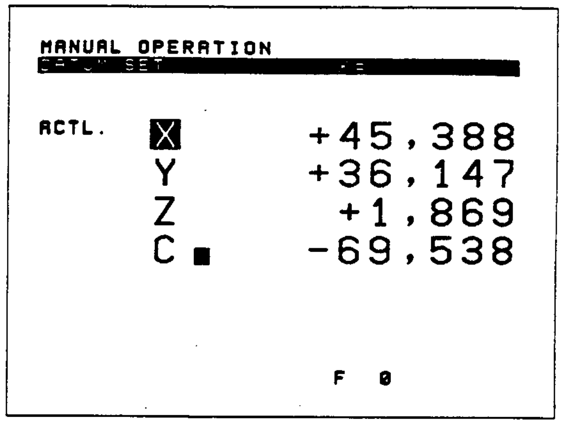
	Cause and behavior of the control:	Remedy:
Input range exceeded	The permissible input range has been exceeded (e. g. excessive feed rate). The value is not accepted and the error message appears.	Clear the value with the "CE" key, enter the correct value and confirm entry.
Incompatible/ contradictory input	For example: L X+50 X+100 During execution of a "TEST" or during program run the TNC stops before the erroneous block, gives an error message and shows number of the block in which the error was found.	Change to the "Programming and editing" operating mode. The error can usually be found either in the displayed block number or in a previously executed block. Fix the error, switch to the "Full sequence" mode and start program run again.
Machine or control fault	Faults which endanger functional stability are indicated with a blinking error message. Write the error message down!	Switch off the machine or control. Correct the fault if possible. Try to restart the program. If the program runs, the fault was transient. If the same error message is displayed, call the repair service of your machine tool manufacturer.

Machine operating modes

Manual operation

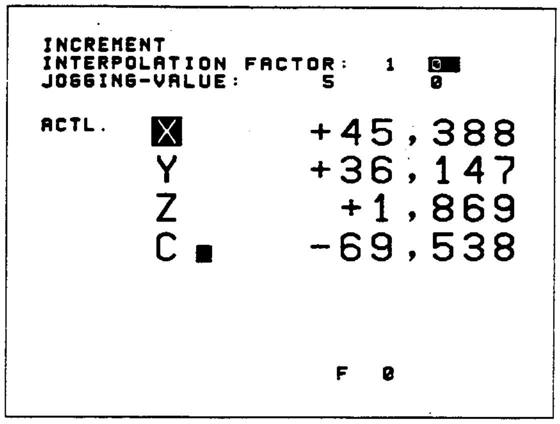
The axes can be moved either with the electronic handwheel or with the machine axis direction buttons.

The position displays can be set to any desired values (datum setting and presetting for machining).



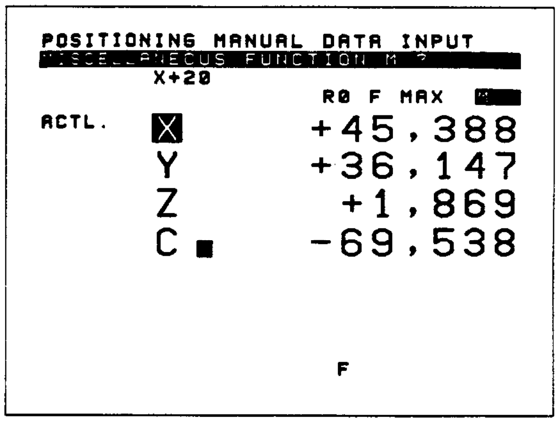
Electronic Handwheel

The axes can be moved either via an electronic handwheel or via the machine axis direction buttons. It is also possible to position by defined jog increments.



Positioning with manual data input (MDI)

The axes are positioned paraxially according to the incremental or absolute data keyed in. These data are not stored.



Program run

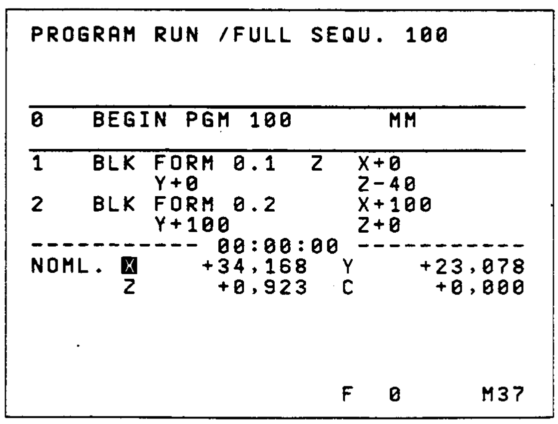
A part program in the memory of the control is executed by the machine.

Full sequence

After starting via the machine START button, the program is automatically executed until the end or a STOP is reached.

Single block

Each block is started separately with the machine START button.



Programming modes

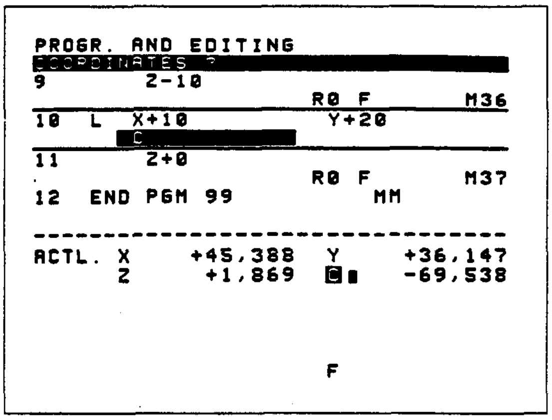
Programming and editing

Part programs can be entered, looked over and altered in the "Programming and editing" operating mode.

External data transfer

EXT

In the "Programming and editing" mode of operation it is possible to read-in and read-out programs via the RS-232-C/V.24 interface.

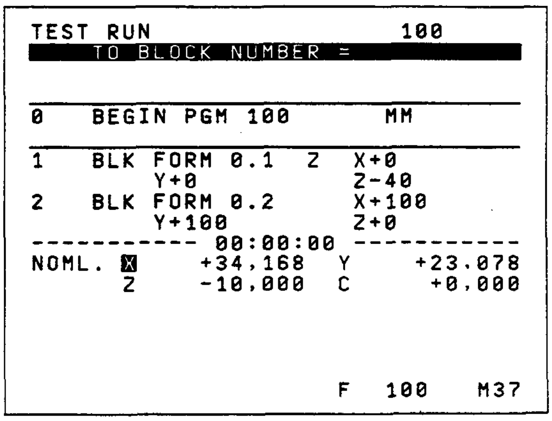


Test run

In the "Test run" operating mode, machining programs are analyzed for logical programming errors, e.g. exceeding the traversing range of the machine, redundant programming of axes, certain geometrical incompatibilities etc.

Note

The programming modes can be used immediately after switch-on.
There is no need to first pass over the reference marks.



Graphic simulation of workpiece machining

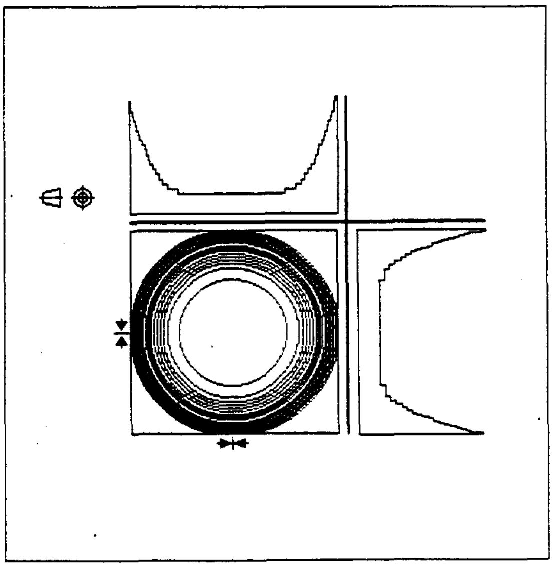
GRAPHICS

BLK FORM MAGN START

MOD

In the "Program run" operating modes "full sequence" and "single block" as well as in "Test run" mode, you can graphically simulate machining programs via the "GRAPHICS" keys.

Display modes:
- plan view with depth indication
- view in three planes
- 3-D view



Accessories
FE 401 Floppy Disk Unit
Data Transfer Software

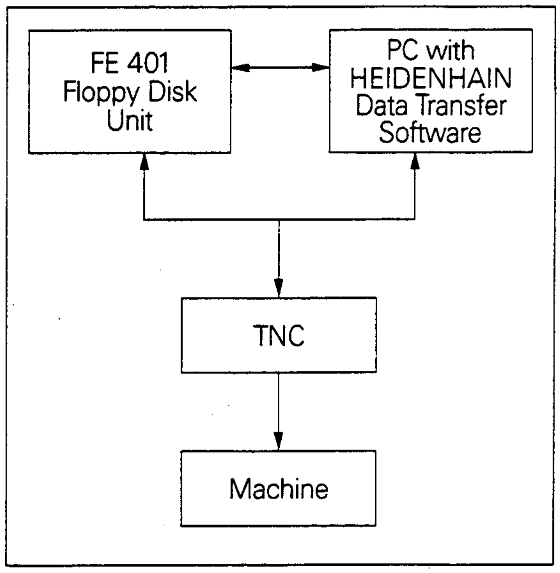
FE 401 Floppy Disk Unit

Part programs which do not have to reside permanently in the control memory can be stored with the FE 401 Floppy Disk Unit.

The storage medium is a normal 3 1/2 inch diskette, capable of storing up to 256 programs and a total of approximately 25 000 program blocks.

Programs can be transferred from the TNC to diskette or vice-versa.

Programs written at off-line programming stations can also be stored on diskette with the FE 401 and read into the control as needed.



A second diskette drive is provided for backing up stored programs and for copying purposes.

Specifications

	FE 401 Floppy Disk Unit with two drives
Data medium	3 1/2 inch diskette, double-sided, 135 TPI
Storage capacity	approx. 790 KB (25 000 blocks); max. 256 programs
Data interface	Two RS-232-C/V.24 data interfaces
Transfer rate	"TNC" interface: 2400/9600/19 200/38 400 baud "PRT" interface: 110/150/300/600/1200/2400/4800/9600 baud

Data transfer software

HEIDENHAIN offers the following data transfer software:

TNC.EXE: For blockwise transfer from the TNC to a PC and vice versa.
FE.EXE: For formatting floppy disks for the FE 401 floppy disk unit, for copying and erasing programs.
FDE.EXE: For data transfer from the FE 401 to a PC and vice versa.

Specifications

HEIDENHAIN data transfer software can be run on DOS-compatible PCs.

Accessories
HR 130/HR 330 Electronic Handwheels

Handwheel

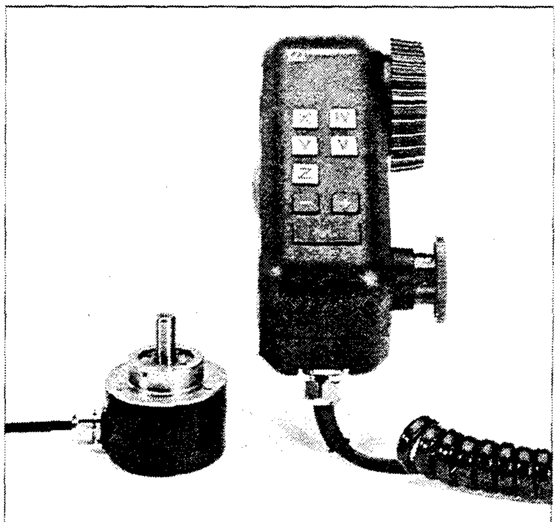
The control can be equipped with an electronic handwheel for better machine setup. Two versions of the electronic handwheel are available:

HR 130

The HR 130 electronic handwheel is designed to be incorporated into the machine control unit. The axis of control is selected at the machine control panel.

HR 330

The portable HR 330 electronic handwheel includes keys for axis selection, axis direction, rapid traverse and emergency stop.



HR 130 HR 330

MOD Functions

MOD

In addition to the main operating modes, the TNC has supplementary operating modes or so-called MOD functions. These permit additional displays and settings.

Selecting

Initiate the dialog MOD

POSITION DATA	↓ ↑	Select MOD functions either via arrow keys
	MOD	or via the MOD key (only paging forward possible).

Terminating

LIMIT X+ = + 350.000	END	Terminate supplementary operating mode.

Transfer numerical inputs with the "ENT" key before terminating the MOD functions.

Vacant memory

The number of free characters in the program memory is displayed with the MOD function "VACANT MEMORY".

Baud rate

The transfer rate for the data interface is specified with "BAUD RATE".

RS-232-C interface

The data interfaces can be switched via "RS-232-C interface" to the following operating modes with the "ENT" key:

- ME operation
- FE 1 operation
- EXT operation: operation with other external devices.

NC software number

The software number of the TNC control is displayed with this MOD function.

PLC software number

The software number of the integrated PLC is displayed with this MOD function.

User parameters

Up to 16 machine parameters can be accessed by the machine operator with this MOD function. These user parameters are defined by the machine manufacturer – he may be contacted for more information.

Code number

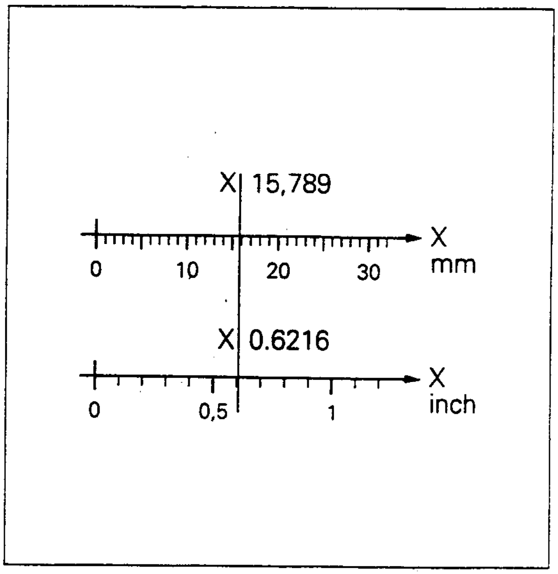
A code number can be entered with this MOD function:

- **123:** select the user parameters.
  These user parameters are accessible on all controls (see User parameters).

Change mm/inch

The MOD function "Change mm/inch" determines whether the control displays positions in the metric system (mm) or in the inch system. You switch between the mm and inch systems via the "ENT" key. After pressing this key the control switches to the other system.

You can recognize whether the control is displaying in mm or inches by the number of digits behind the decimal point:
X15.789 mm display
X 0.6216 inch display.



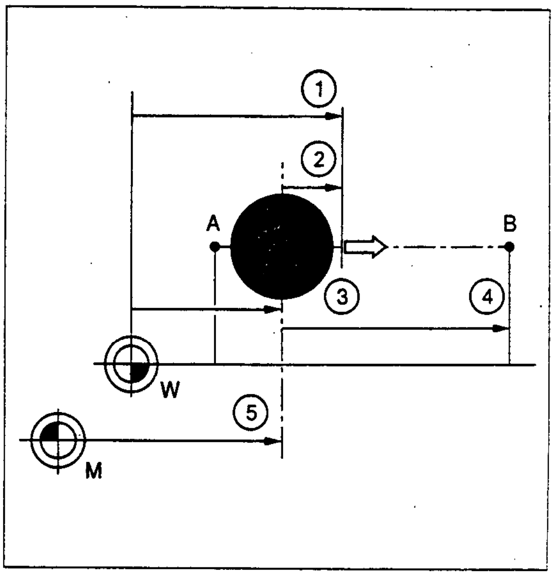
Position displays

The following position displays can be selected:

① nominal position of the control	NOML
② difference nominal/actual position (lag distance)	LAG
③ actual position	ACTL.
④ remaining distance to programmed position	DIST.
⑤ position based on the machine datum	REF

A = last programmed position (starting position)
B = new (programmed) target position, which is presently targeted
W = Workpiece datum for the part program
M = Machine datum

Switchover is with the "ENT" key.



MOD Functions
Traverse range limits

MOD

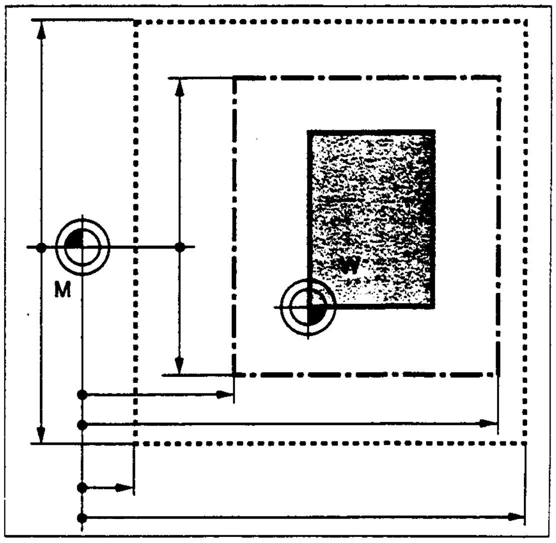
Limits

The maximum displacements are preset by fixed software limits.

The MOD function "Limits" enables you to specify additional software limits for a "safety range" within the limits set by the fixed software limits.

Thus you can, for example, protect against collision when clamping a dividing attachment. The displacements are limited on each axis successively in both directions based on the **machine datum** (reference marks). The position display must be switched to REF before specifying the limit positions of the position display.

To work without safety limits, enter the maximum values +30 000.000 or −30 000.000 [mm] for the corresponding axes.



Effectiveness

The entered limits do not account for tool compensations. Like the software limit switches, they are only effective after you traverse the reference points. They are reactivated with the last entered values after a power interruption.

Determine values

To determine the input values, switch the position display to REF.	X Y Z	Traverse to the end positions of the axis/axes which is/are to be limited. Note the appropriate REF displays (with signs).

Enter values

Select — MOD ↓ Continue pressing until LIMIT appears.

Enter the limit(s)	☐ ENT	Enter value, or
	↓	select the next limit
	⋮	
	END	terminate the input.

User Parameters
General Information

MOD

Machine parameters

The TNC contouring controls are individualized and adapted to the machine via machine parameters (MP). These parameters consist of important data which determine the behavior and performance of the machine.

Parameters accessible for the user

Certain machine parameters which determine functions dealing only with operation, programming and displays are accessible for the user.

Examples

- Scaling factor only effective on X, Y or on X, Y, Z.
- Adapting the data interface to different external devices.
- Display possibilities of the screen.

Accessibility

The user can access these machine parameters in two ways:

- Access by entering the **code number 123.**
  This access is possible on every control (see code number 123).
- Access to additional parameters via the MOD function **User parameters.**
  You can only access via the MOD function if the manufacturer has made the machine parameters accessible for this purpose.

The machine manufacturer can inform you about the sequence, meaning, texts etc. of user parameters.

Only these machine parameters may be changed by the user. In no case should the user change any non-accessible machine parameters.

Selection	MOD ↑ ENT Select the user parameter.
	↓ Continue pressing until the desired USER PARAMETER or dialog appears.
	Enter numbers.
	END END Terminate or select further user parameters with ↓ and then terminate.

After entering the code number **123** via MOD, the following machine parameters and the parameters for the data interface (see index "Programming Modes", "External data transfer") can be selected and changed.

Feed rate

Function	Parameter number	Input	Input values
Feed rate, if not specified in the NC program	1090	0 to 30000 mm/min	
Positioning speed during flushing and short circuit	1091	0 to 30000 mm/min	
Maximum speed of circular movement during the disk cycle Mode: 0 and 4 (erosion) Mode: 0 and 4 (free runing) Mode: 1 and 5 (erosion) Mode: 1 and 5 (free runing) Mode: 2 and 6 (erosion) Mode: 2 and 6 (free runing)	 1092 1093 1094 1095 1096 1097	0 to 30000 mm/min	

Eroding

Function	Parameter number	Input	Input values
Value for the Q parameter Q157, if during the tool definition with TOOL DEF the question about the following electrode was answered with "no".	2040	0 to 10	
Starting position for re-approaching the contour after a short circuit or power interruption	2050	0 to 2 mm	
Starting position for re-approaching the contour after flushing	2051	0 to 2 mm	
Positioning with free-run feed rate during erosion	2060	1 to 500 mm/min	
Rotational speed of the C axis if M3/M4 was programmed in the NC program	2090	0 to 100 rpm	
Duration of free-run signal so that control can complete an eroding procedure at the end position	2110	0 to 99.9 s	
Arc recognition	2120	1 to 99.9 s	
Constant contouring speed during flushing	2190	0 → Electrode is stopped after every NC block 1 → If the programmed contour is geometrically continuous, the electrode moves with constant speed 2 → Electrode always moves with constant speed	

Measuring with the TOUCH PROBE

Function	Parameter number	Input	Input values
Feed rate for probing	6120	80 to 3000 mm/min	
Measuring distance	6130	0 to 30000.000 mm	
Rapid traverse for probing	6150	80 to 30000 mm/min	

Display and programming

Function	Parameter number	Input	Input values
Programming station	7210	0 → Control 1 → Programming station: PLC active 2 → Programming station: PLC inactive	
Switching of dialog language German/English	7230	0 → First dialog language 1 → Second dialog language (English)	
Inhibit PGM input for PGM no. = OEM cycle no.	7240	0 → Inhibited 1 → Uninhibited	
Display of the current feed rate before start in the **manual operating modes** (same feed rate in all axes, i.e. smallest programmable feed rate)	7270	0 → No display 1 → Display	
Feed rate display	7271	0 → Display 1 → No display	
Display of current machining time	7272	0 → Display 1 → No display	
Decimal character	7280	0 → Decimal comma 1 → Decimal point	
Clearing the status display and the Q parameters with M02, M30 and end of program	7300	0 → Status display is not cleared 1 → Status display is cleared	
Graphics (display mode) Switch over projection type "display in 3 planes" Rotate the coordinate system in the machining plane by 90°	7310 Bit 0 1	 + 0 → European preferred + 1 → American preferred + 0 → No rotation + 2 → Coordinate system rotated by +90°	

Machining and program run

Function	Parameter number	Input	Input values
"Scaling" cycle is effective on 2 axes or 3 axes	7410	0 → 3 axes 1 → in the machining plane	
Output of M functions Programmed stop at M06 Output of M89, modal cycle call Axis stop with output of M function **Exceptions:** Axis stop occurs with all M functions which result in a programmed stop (e.g. M00, M02) or with a STOP or CYCL CALL block.	7440 Bit 0 1 2	 + 0 → Programmed stop at M06 + 1 → No programmed stop at M06 + 0 → No cycle call, normal output of M89 at start of block + 2 → Modal cycle call at end of block + 0 → axis stop + 4 → no axis stop	
Constant path speed at corners	7460	0 to 179.999	

Hardware

Function	Parameter number	Input	Input values
Feed rate override	7620 Bit		
Feed rate override if rapid traverse key is pressed in operating mode "Program run"	0	+ 0 → Override inactive + 1 → Override active	
Feed rate override in 2 % increments or 1 % increments	1	+ 0 → 2 % increments + 2 → 1 % increments	
Feed rate override if rapid traverse key and external direction buttons are pressed	2	+ 0 → Override inactive + 4 → Override active	

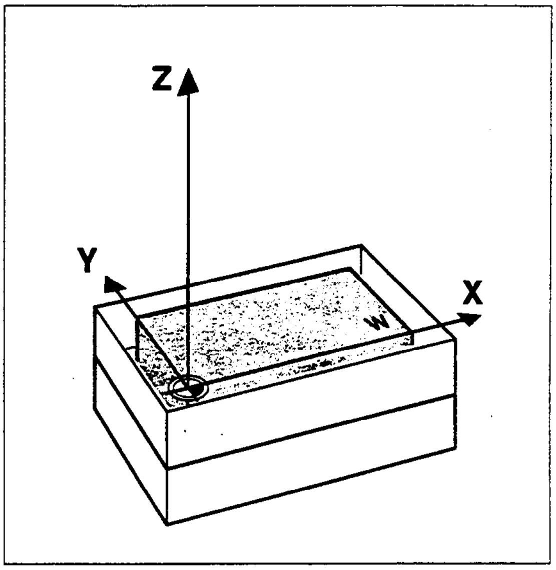
Coordinates
Coordinate system

In a part program, the **nominal positions** of the electrode are defined in relation to the workpiece; encoders on the machine axes continuously deliver the signals needed by the control for determining the current **actual position.**

A reference system is always required for determining position. In the present case, such a system must be **workpiece-based.**

Cartesian coordinates

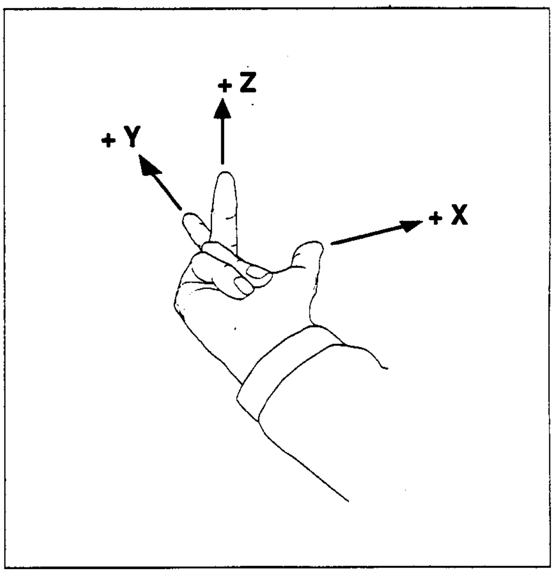
The reference system normally used is the **rectangular** or **Cartesian* coordinate system** (coordinates are those values which define a unique point in a reference system). The system consists of three coordinate axes, perpendicular to each other and lying parallel to the machine axes, which intersect each other at the so-called origin or (absolute) zero point. The coordinate axes represent mathematically ideal straight lines with divisions; the axes are termed X, Y and Z.



Right-hand rule

You can easily remember the traversing directions with the **right-hand rule:** the positive direction of the X axis is assigned to the thumb, that of the Y axis to the index finger, and that of the Z axis to the middle finger.

ISO 841 specifies that the **Z axis** should be defined according to the **direction of the tool axis,** whereby the positive Z direction always points **from the workpiece to the electrode.**

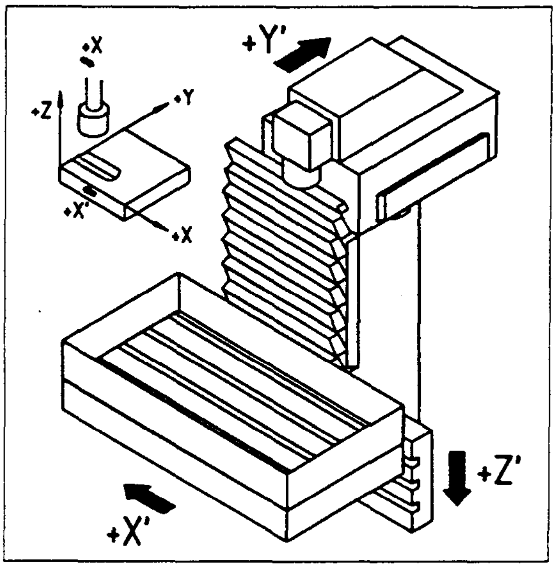


*) after the French mathematician René Descartes, in Latin Renatus Cartesius (1596 – 1650).

Coordinates
Datum

Relative tool movement

Part programs are always written with workpiece-based coordinates X, Y, Z. That is, they are written as if the tool moves and the workpiece remains still, independent of the machine type.

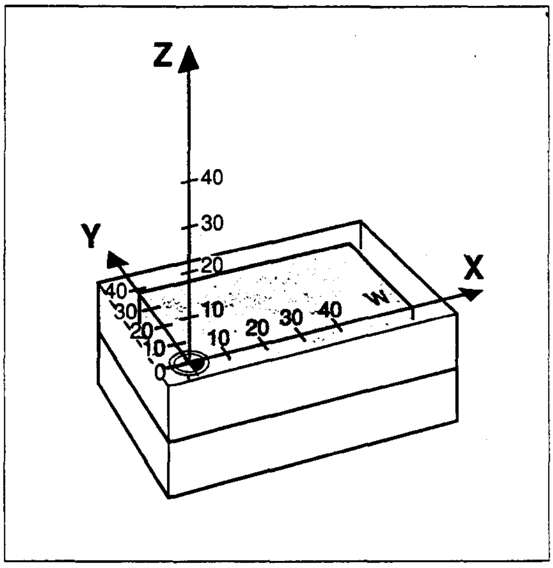


If, however, the work support on a given machine actually moves in any axis, then the direction of the coordinate axis and the direction of traverse will be opposite.

In such a case the machine axes are designated as X', Y' and Z'.

Zero point of the coordinate system

For the **zero point of the coordinate system,** *the position on the workpiece which corresponds* to the datum of the part drawing is generally chosen – that is, the point to which the part dimensioning is referenced.



For reasons of safety, the workpiece datum in the Z axis is almost always positioned at the highest point on the workpiece.

The datum position indicated in the drawing to the right is valid for all programming examples in *this manual.*

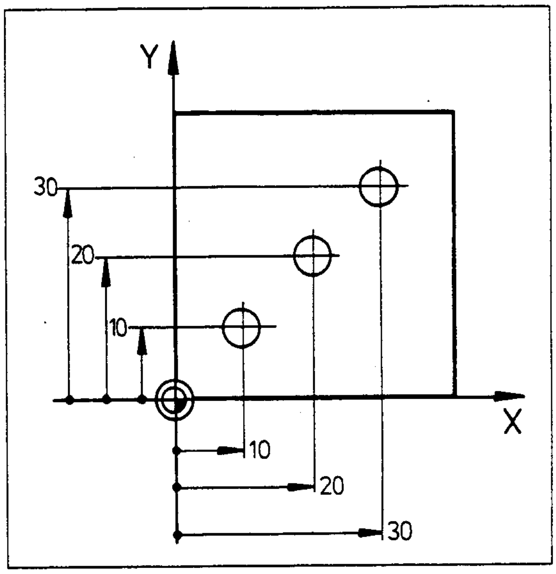
Machining operations in a horizontal plane require freedom of movement mainly in the positive X and Y directions. Infeeds starting from the upper edge of the workpiece Z = 0 correspond to negative position values.

Datum Setting

The workpiece-based rectangular **coordinate system** is defined when the coordinates of any **datum P** are known – that is, when the tool is moved to the datum position and the control "sets" the corresponding coordinates (datum setting).

Coordinates
Absolute and incremental coordinates

If a given point on the workpiece is referenced to the **datum,** then one speaks of **absolute coordinates** or absolute dimensions. It is also possible to indicate a position which is referenced to **another known workpiece position:** in this case one speaks of **incremental coordinates** or incremental dimensions.



Absolute dimensions

The machine is to be moved **to** a certain position or **to** certain nominal coordinates.

Example: X+30 Y+30

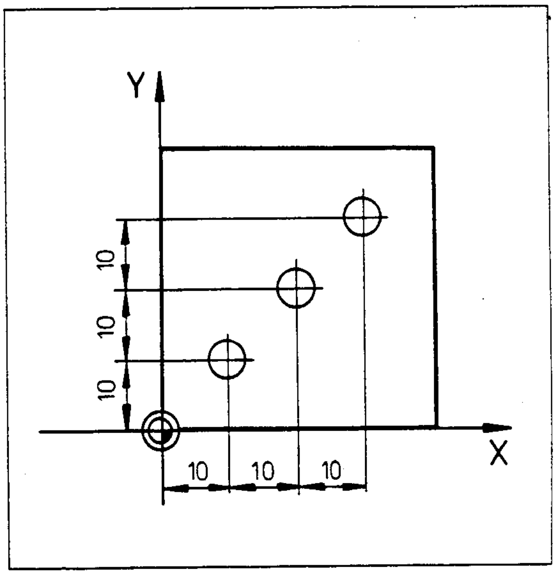
Dimensions in this manual are given as **absolute Cartesian dimensions** unless otherwise indicated.

Incremental dimensions

Incremental dimensions in a part program always refer to the **immediately preceding nominal position.** Incremental dimensions are indicated by the letter I.

The machine is to be moved **by** a certain distance: it moves from the previous position along a distance given by the incremental nominal coordinate values.

Example: IX+10 IY+10



Mixing absolute and incremental dimensions

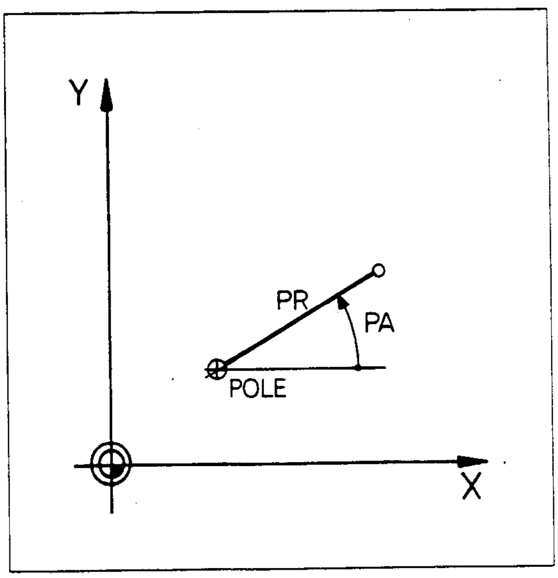
It is possible to mix absolute and incremental coordinates within the same program block.

Example: L IX+10 Y+30

Polar coordinates

Positions on the workpiece can also be programmed by entering the radius and the direction angle referenced to a pole (see index Programming Modes, Polar coordinates).

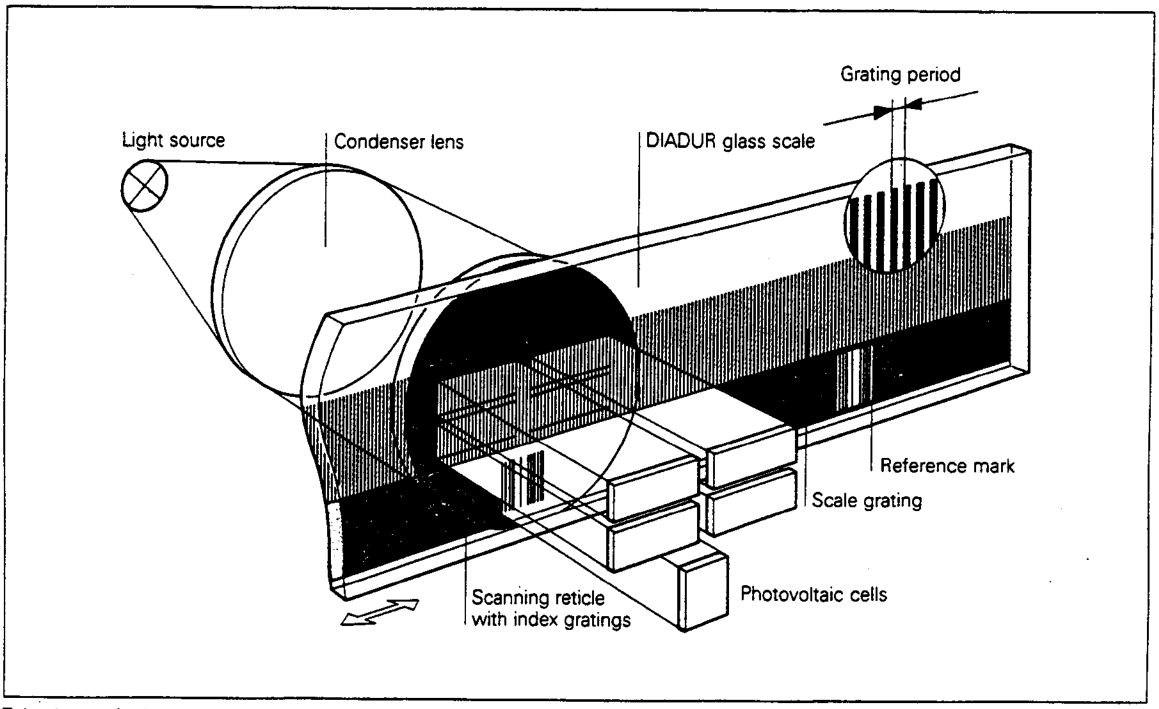
CC = Pole
PR = Polar radius (distance from pole)
PA = Polar angle (direction angle)



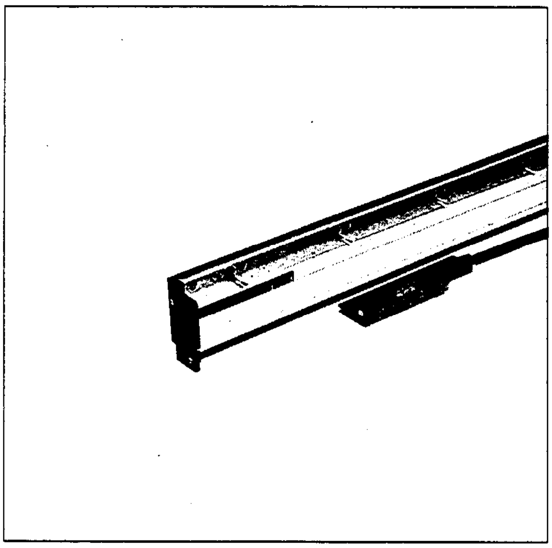
Coordinates
Linear and angle encoders

Linear and angle encoders in machine tools

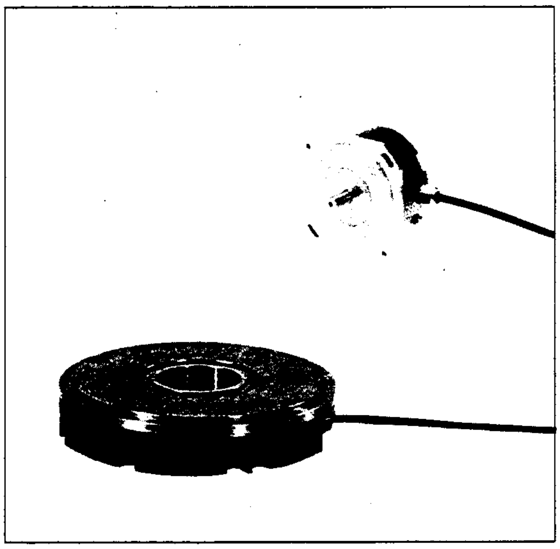
Each machine axis requires a measuring system to provide the control with information on the actual position: linear encoders for linear axes, angle encoders for rotary axes.



Principle of photoelectric scanning of fine gratings



LS 103 C



RON 706 C, ROD 250 C

With **linear axes,** position measurement is generally based on either

- a photoelectrically scanned **steel or glass scale,** or
- the **high-precision ballscrew,** which also functions as a drive element (the electrical signals are then produced by a rotary encoder coupled to the ballscrew).

With **rotary axes,** a graduated disk permanently attached to the axis is photoelectrically scanned. The TNC forms the position value by counting the generated impulses.

Coordinates
Linear and angle encoders

Linear and angle encoders are **machine-based:**

Datum

The datum for determination of the nominal and actual position must correspond to the workpiece datum, or be brought into correspondence by **setting** the correct position value (= the position value determined by the workpiece datum) in any axis position. This procedure is called datum setting (or datum presetting).

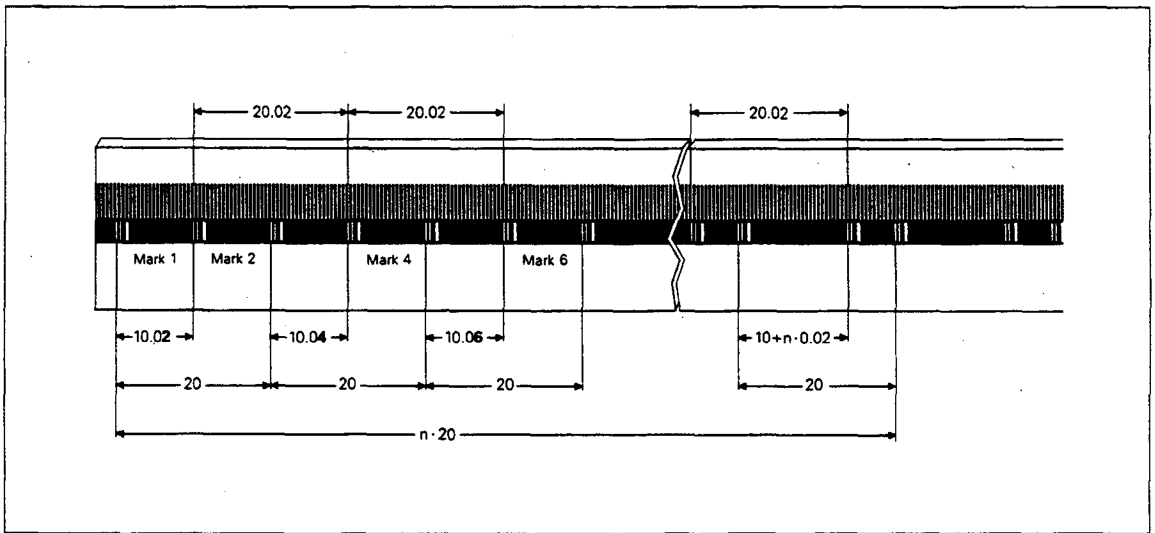
Reference marks

After the control has been switched off or after a power interruption, it is necessary to set the datum again. To simplify this task, the encoders possess **reference marks,** which in a sense also represent datum points.

The relationship between axis positions and position values which were established by the last setting of the workpiece datum (datum setting), are **automatically** retrieved by traversing over the reference marks after switch-on. This also re-establishes the machine-based references such as the software limit switch or tool change position.

In the case of linear encoders with distance-coded reference marks, the machine axes need only be traversed by a maximum of 20 mm. For angle encoders with distance-coded reference marks, a rotation of just 20° is required.

Linear encoders with only one reference mark have an **"RM"** label which indicates the position of the reference mark, while angle encoders with one reference mark indicate the position with a notch on the shaft.



Schematic of scale with distance-coded reference marks

Notes

Machine Operating Modes (M)

Switch-On
Traversing the reference points

Switch-On

	Switch power on.

MEMORY TEST	The TNC tests the internal control electronics. The display is automatically cleared.
POWER INTERRUPTED	CE Delete the message. The control then tests the EMERGENCY STOP circuit.

RELAY EXT. DC VOLTAGE MISSING	I Switch on the control DC voltage.

MANUAL OPERATION TRAVERSE REFERENCE POINTS Z AXIS X AXIS Y AXIS 4th AXIS	START Traverse the axes over the reference points in the displayed sequence. Start each axis separately or X Y Z . . . move the axes with the external direction keys. The sequence of the axes is determined by the machine manufacturer.

MANUAL OPERATION	"Manual operation" is now selected automatically.

Handwheel

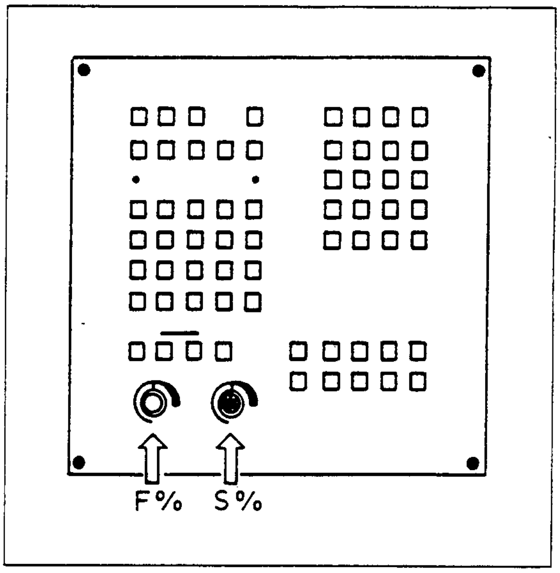
The reference points can also be traversed by using the handwheel.

Encoders

The required traversing distance for linear and angle encoders with distance-coded reference marks is max. 20 mm or 20°. If the encoder has only one reference mark, it must be traversed after switch-on.

Manual Operation
Traversing with the axis direction buttons/ Handwheel/Rotational speed of the C axis

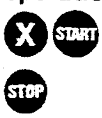
The machine axes can be moved and the datum set in the "Manual" operating mode.



Jog mode

The machine axis moves as long as the corresponding external axis direction button is held down. Several axes can be driven simultaneously in the jog mode.

Continuous operation



If the machine "START" button is pressed simultaneously with an axis direction button, the selected machine axis continues to move after the two buttons are released. Movement is stopped with the machine "STOP" button.

Handwheel

Traversing with the handwheel is also possible in the "Manual" mode of operation. The distance of traverse per revolution is set through the interpolation factor (see index "Machine Operating Modes", "Electronic Handwheel").

Feed rate override

The traverse speed (feed rate) is preset by machine parameters and can be varied with the feed rate override (F %) of the control.

Rotational speed of the C axis for free rotation

The rotational speed of the C axis can be changed through the general user parameter **2090** (accessible through the code number 123).

The erosion parameter for the rotational speed of the C axis is activated in the "Single block" or "Full block" program run modes of operation.

S Override

The rotational speed of the C axis during free rotation (M03 or M04) can be changed with the aid of the S override.

Manual Operation
Miscellaneous functions M

Miscellaneous function M

Use the "STOP" key to enter a miscellaneous function:

Initiate the dialog STOP

MISCELLANEOUS FUNCTION M ?	Enter the M function.
	ENT Confirm entry.
	START Activate the miscellaneous function.

Miscellaneous functions with predetermined function:

M	Function	Active at block beginning	end
03	Free rotation of C axis, clockwise	●	
04	Free rotation of C axis, counterclockwise	●	
05	Stop free rotation of the C axis		●
08	Flushing on	●	
09	Flushing off		●
36	Erosion on	●	
37	Erosion off	●	
38	Transfer values from the datum table to Q parameters (Q81 to Q84)	●	
39	Transfer values from Q parameters (Q81 to Q84) to datum tables	●	

Setup
Datum setting with probe functions

The "TOUCH PROBE" function offers considerable benefits when used together with the short circuit test signal. One is that the workpiece does not have to be aligned precisely to the machine axes: The TNC will determine and compensate misalignment automatically ("basic rotation"). Another important benefit is significantly faster and more accurate datum setting.

Probing functions

TOUCH PROBE

The touch probe functions described below can be employed in the "Manual operation" and "Electronic handwheel" operating modes.

Pressing the "TOUCH PROBE" key calls the menu shown here to the right. The probing function is selected with the cursor keys and entered with the "ENT" key.

CALIBRATION EFFECTIVE LENGTH
CALIBRATION EFFECTIVE RADIUS
BASIC ROTATION
SURFACE = DATUM
CORNER = DATUM
CIRCLE CENTER = DATUM

Calibration

The effective length of the electrode and the effective radius can be calibrated before beginning touch probe work. Both dimensions are determined by CALIBRATION routines, stored in the control and calculated into all the following probe values.

Terminating the probing functions

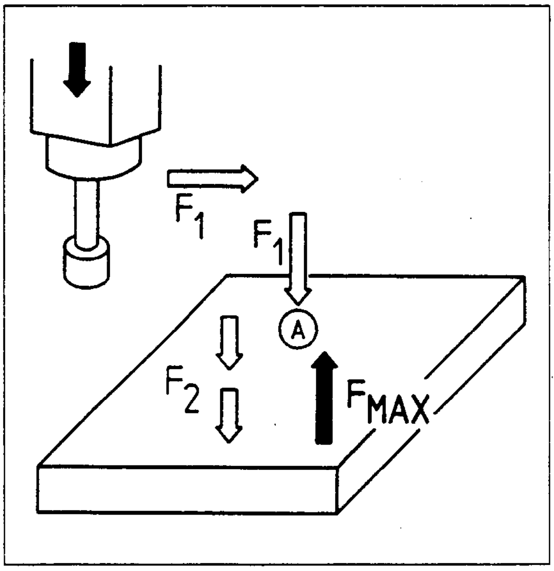
The probing functions can be terminated with "END □".

Calibrating/ working procedure

The electrode traverses to the side or upper surface of the work. The feed rate during measurement and the maximum measuring distance are set by the machine manufacturer via machine parameters.

A short circuit signals to the control that the electrode has made contact with the workpiece. The control stores the coordinates of the contacted points. The probing axis is stopped and retracted to the starting point. Overrun caused by braking does not affect the measured result.

Ⓐ = pre-positioning with the external axis direction buttons.
F1 = feed rate for pre-positioning.
F2 = feed rate for probing.
FMAX = retraction in rapid traverse.

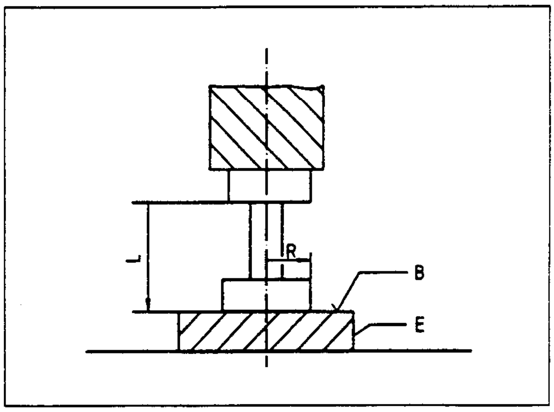


Setup
Calibrating effective length

Work aid: ring gauge

For calibration of the effective length, a ring gauge of known height and known internal radius is clamped to the machine table.

G = ring gauge
D = datum plane (surface)
L = length of the electrode
R = electrode radius

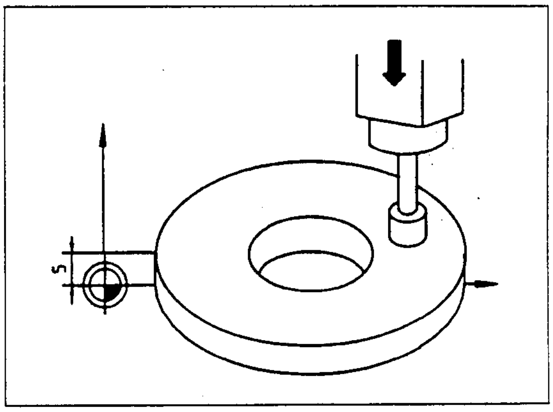


Procedure

The reference plane is set with the electrode prior to calibration.

To determine the effective length of the electrode, the electrode touches the datum plane. After contacting the surface, the electrode is retracted in rapid traverse to the starting position.

The length L is stored by the control and automatically compensated during the measurements.



Initiate the dialog — TOUCH PROBE

CALIBRATION EFFECTIVE LENGTH — ↑ ENT Select probing function and enter.

TOOL AXIS = Z	☐ Enter a different tool axis if required.
	↓ Select the "Datum".
DATUM +5	☐ ENT Enter the datum in the tool axis, e.g. +5.0 mm.

Z+ Z–	X Y Z Move the electrode to the vicinity of the reference plane. ← → Select the direction of electrode movement, here Z–. START The electrode moves in negative Z direction. After touching the surface and returning to the starting position, the control automatically switches to the "Manual operation" or "Handwheel" operating mode.

Display

The value for effective length can be displayed by selecting "Calibration effective length" again.

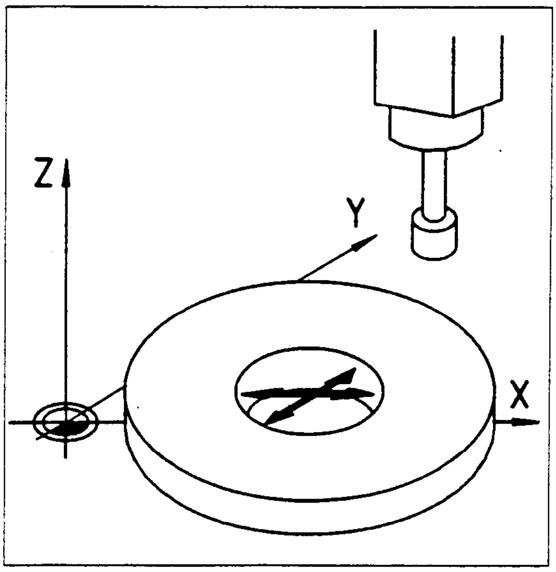
3D Touch Probe
Calibrating effective radius

Procedure

The electrode is lowered into the bore of the ring gauge. 4 points on the wall must be touched to determine the effective radius of the electrode. The traverse directions are determined by the control, e.g. X+, X–, Y+, Y– (tool axis = Z).

The electrode is retracted in rapid traverse to the starting position after every deflection.

The radius R is stored by the control and automatically compensated during the measurements.



Initiate the dialog — TOUCH PROBE

CALIBRATION EFFECTIVE RADIUS — ↑ ENT Select probing function and enter.

TOOL AXIS = Z	Enter another tool axis if required.
	↓ Select "Radius ring gauge".
RADIUS RING GAUGE = 10	ENT Enter the radius of the ring gauge, e.g. 10.0 mm.

X+ X– Y+ Y–	X Y Z Traverse approximately to the center of the ring gauge. ← → Select the traversing direction of the electrode (only necessary if you prefer a certain sequence or the exclusion of one probing direction). START Probe a total of 4 times. After contacting the wall of the ring gauge four times, the control automatically switches to the "Manual operation" or "Handwheel" operating modes.

Display

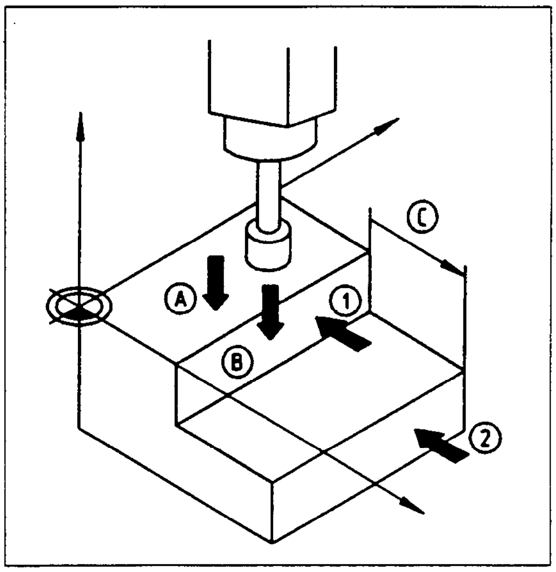
You can display the value for effective radius by selecting "Calibration effective radius" again.

Error messages

TOUCH POINT INACCESSIBLE
The short circuit signal did not occur within the measuring distance (machine parameter).

Setup
Reference surface, Position measurement

The position of a surface on the clamped workpiece is determined with the probing function "Surface = datum".



Functions

- Setting the reference plane Ⓐ
- Measuring positions Ⓑ
- Measuring distances Ⓒ

Measuring distances

You can also measure distances on an aligned workpiece.

- Probe the first position ① and set the datum (e.g. 0 mm).
- Probe the second position ②.
  The distance can be read in the "Datum" display.

Measuring positions

Initiate the dialog — TOUCH PROBE

SURFACE = DATUM — ↑ ENT Select probing function and enter.

X+ X– Y+ Y– Z+ Z– C+ C–	X Y Z Move to the starting position. ← → Select the traversing direction, e.g. Z–. START Move the electrode in negative Z direction. The electrode is retracted in rapid traverse to the starting position after touching the surface.

Measured value

DATUM NUMBER 1	Z+1,804	The measured value can be stored in the datum table as a datum or in the "Tool definition" cycle as a compensation value.
DATUM	Z+1,804	
COMPENSATION VALUE	Z+1,804	

	↓ ↑ Select the desired transfer function. Enter new value, if necessary. ENT Confirm entry.

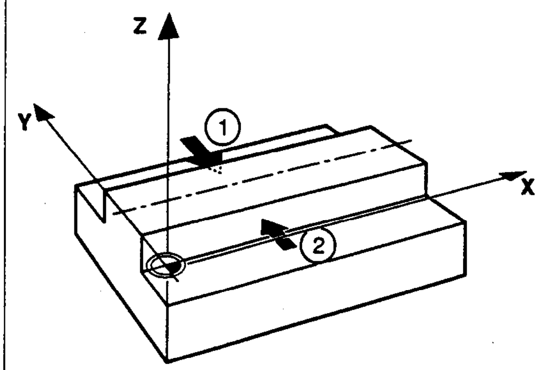
The "DATUM NUMBER" function for transferring the measured value to the datum table is described in the section "Touch points in the datum table 0.D".

The "CORRECTION VALUE" function for transferring the measured value in the "TOOL DEF." cycle is described in the section "Touch points in Cycle 3: Tool Definition".

Setup
Workpiece midpoint = Datum

Procedure

The "Workpiece midpoint = Datum" function can be used to calculate the workpiece midpoint (M) after probing two sides ① and ② of the workpiece.



Initiate the dialog

TOUCH PROBE

WORKPIECE MIDPOINT = DATUM

Select probe function and enter.

X+ X– Y+ Y– Z+ Z– C+ C–	X Y Z Move the electrode to the starting position ①. Select the probing direction, e.g. X+. START The electrode moves in the selected direction. After touching the workpiece, it returns to the starting position.

	X Y Move the electrode to the starting position ②. START The electrode moves in the selected direction, e.g. X–. After touching the workpiece, the electrode returns to the second starting position.

Workpiece midpoint

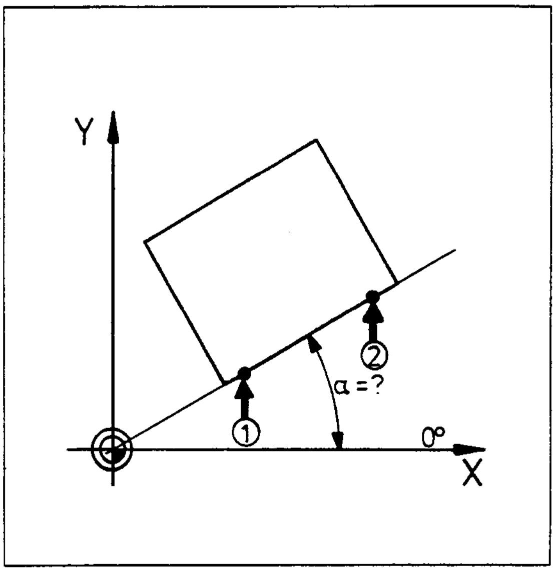
DATUM NUMBER 1	Z+1.804	The workpiece midpoint can be stored in the datum table as a datum or in the "Tool definition" cycle as a compensation value.
DATUM	Z+1.804	
COMPENSATION VALUE	Z+1.804	

	Select the desired transfer function. Enter new value, if necessary. ENT Confirm entry.

The "DATUM NUMBER" function for transferring the measured value to the datum table is described in the chapter "Touch points in the datum table 0.D".
The "CORRECTION VALUE" function for transferring the measured value in the "TOOL DEF." cycle is described in the chapter "Touch points in Cycle 3: Tool Definition".

Setup
Basic rotation, Angular measurement

The probing function "Basic rotation" determines the angle of deviation of a plane surface from a nominal direction. The angle is determined in the machining plane.



Functions

- Basic rotation (the control compensates for an angular misalignment)
- Correct an angular misalignment (on a machine with rotary axis)
- Measure an angle.

Basic rotation

Initiate the dialog — TOUCH PROBE

BASIC ROTATION — ↑ ENT Select probing function and enter.

ROTATION ANGLE = 0	↓ Select the "Rotation angle".
	ENT Enter the nominal direction of the surface to be probed, e.g. 0°.

X+ X– Y+ Y–	X Y Z Move the electrode to the starting position ①.
	← → Select the probing direction, e.g. Y+.
	START The electrode travels in the selected direction, e.g. Y+. The electrode returns to the starting position after touching the side surface.

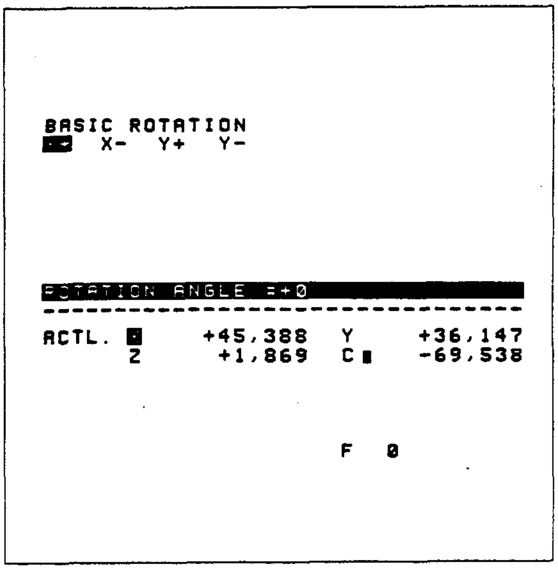
	X Y Move the electrode to the starting position ②.
	START The electrode travels in the selected direction, e.g. Y+. The electrode returns to the second starting position after making contact. The control automatically switches to the "Manual operation" or "Handwheel" operating mode.

Setup
Basic rotation, Angular measurement

Displaying the rotation angle

The measured rotation angle is displayed by selecting the probing function "Basic rotation".

Compensation of angular misalignment is registered on the screen with "ROT" in the status display. It also remains stored after a power interruption.



Cancelling the basic rotation (rotation angle 0°)

The basic rotation is cancelled by selecting the probing function "Basic rotation" and entering a 0° rotation angle.
The "ROT" display is cleared.

Once basic rotation is activated, all subsequent programs are executed with rotation and shown rotated in the graphic simulation.

Measuring angles

In addition to basic rotation, angle measurements can also be performed on aligned workpieces.

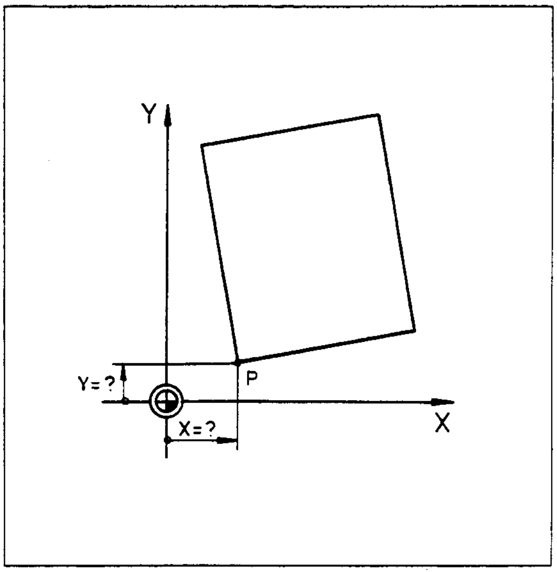
Carry out the following procedure:

- Execute a basic rotation.
- Display the rotation angle.
- Cancel the basic rotation.

TOUCH PROBE

Setup
Corner = datum/ Determining corner coordinates

With the probing function "Corner = datum", the control computes the coordinates of a corner on the clamped workpiece. The computed value can be taken as datum for subsequent machining. All nominal positions then refer to this point.

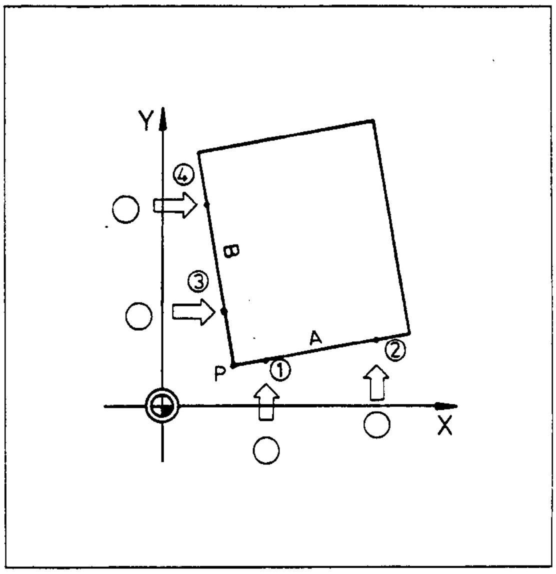


The probing function "Basic rotation" should be performed before "Corner = datum".

Procedure

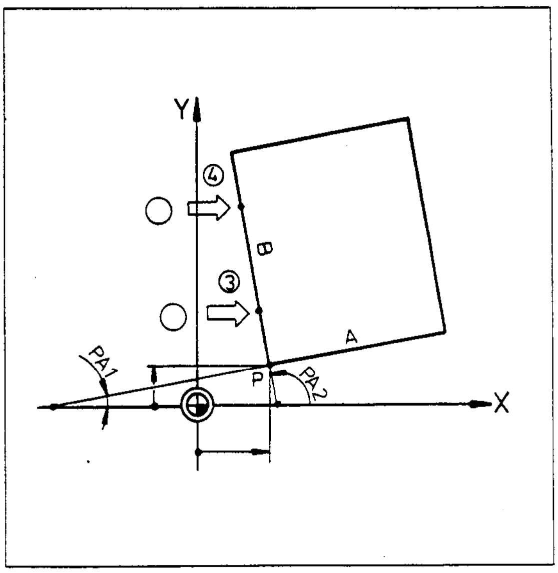
The electrode touches two side surfaces (see figure) from two different starting positions per side.

The corner point P is computed by the control as the intersection of straight line A (contact points ① and ②) with straight line B (contact points ③ and ④).



After performing a basic rotation

If the probing function "Corner = datum" is called after performing a basic rotation (straight line A), the first side need not be contacted.



TOUCH PROBE

Setup
Corner = datum/ Determining corner coordinates

To transfer the direction of the first side face from the routine "basic rotation", simply respond to the dialog query **TOUCH POINTS OF BASIC ROTATION ?** by pressing the "ENT" key (otherwise "NO ENT").

If only the probing function "CORNER = DATUM" is performed, then it does not contain a basic rotation.

Initiate the dialog — TOUCH PROBE

CORNER = DATUM — Select probing function and enter.

First side face

X+ X– Y+ Y–	X Y Z Move the electrode to the first starting position.
	Select the probing direction, e. g. Y+.
	START The electrode travels in the selected direction. After touching the side face, the electrode is retracted to the starting position.

Traverse to the second starting position and probe in the same probing direction as described above.

Second side face

X+ X– Y+ Y–	X Y Z Move the electrode to the third starting position.
	Select the probing direction, e. g. X+.
	START The electrode travels in the selected direction. After touching the side face, the electrode is retracted to the starting position.

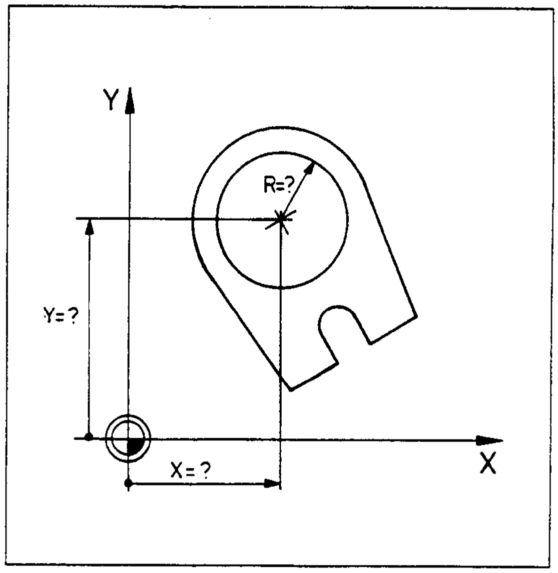
Traverse to the fourth starting position and probe in the same probing direction as described above.

Display corner coordinates/ Setting the datum

DATUM X+0	Enter the corner coordinates for X and Y if required, e. g. X+0, Y+0.
DATUM Y+0	
	ENT Confirm entries.

Setup
Circle center = datum/ Determining the circle radius

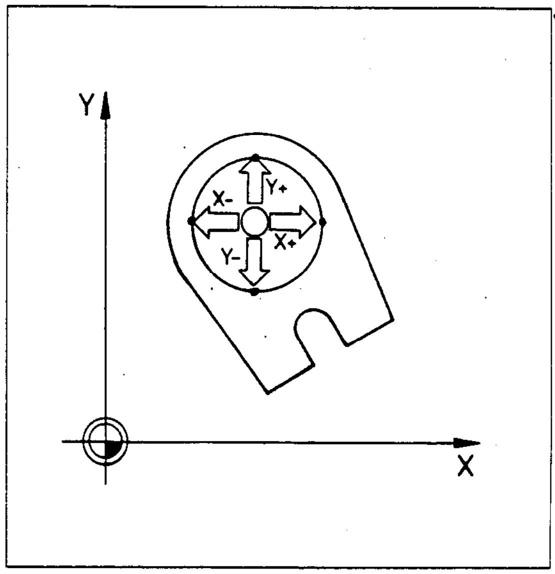
In the probing function "Circle center = datum", the control computes the coordinates of the circle center and the circle radius on a clamped workpiece with cylindrical surfaces. The coordinates of the center can be used as the datum for subsequent machining. All nominal positions are then referenced to this point.



The "Basic rotation" probing function must be carried out prior to "Circle center = datum".

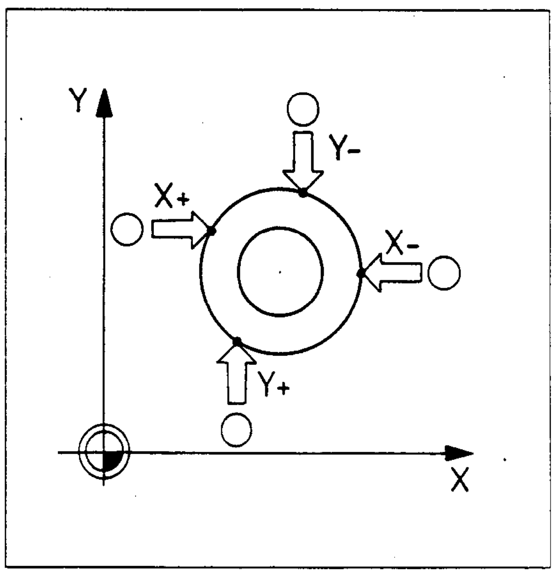
Circular pocket

Position the electrode in the pocket with the remote axis direction keys. Four different positions are then touched by pressing the machine START button.



Outer cylinder

On workpieces with cylindrical outer surfaces, the probing directions must be specified for each of the four points.



TOUCH PROBE

Setup
Circle center = datum/ Determining the circle radius

Initiate the dialog — TOUCH PROBE

CIRCLE CENTER = DATUM — ↓ ENT Select the probing function and enter.

X+ X– Y+ Y–	X Y Z Move the electrode to the first starting position. ← → Select the probing direction if required, e.g. X–. START Electrode travels in the selected direction. After touching face, the electrode is retracted to the starting position.

Traverse to the second and third starting positions and probe in different directions as described above.

X+ X– Y+ Y–	X Y Z Move the electrode to the fourth starting position. ← → Select the probing direction if required, e.g. Y–. START The electrode travels in the selected direction. The electrode is retracted to the starting position after touching the side face.

Display

X+54.3 Y+21.576	Coordinates of the circle center.
PR+20	Circle radius.

Circle center

DATUM NUMBER 2 X+0.317 DATUM NUMBER 2 Y+0.093 DATUM X+0.317 DATUM Y+0.093 COMPENSATION VALUE X+0.317 COMPENSATION VALUE Y+0.093	The circle center can be stored in the datum table as a datum or in the „Tool definition" cycle as a compensation value.

↓ ↑ Select the desired transfer function. Enter new value, if necessary. ENT Confirm entry.

The "DATUM NUMBER" function for transferring the measured value to the datum table is described in the chapter "Touch points in the datum table 0.D".

The "CORRECTION VALUE" function for transferring the measured value in the "TOOL DEF." cycle is described in the chapter "Touch points in Cycle 3: Tool Definition".

TOUCH PROBE

Setup
Touch points in the datum table 0.D

In the "Manual" and "Electronic handwheel" modes of operation, the measured values from the probing functions described below can be transmitted to the datum table 0.D.

Select datum number

Q 80

Initiate dialog Q

Q	=	8 0	Select parameter Q80 to define the datum number.
Q	=+3	ENT	The current datum number is displayed (e.g. 3).
		ENT	Enter the new datum number and confirm entry.

TOUCH PROBE

After selecting the new datum number, you can execute the desired probe function.

Probe functions

The following probe functions provide a menu item for transferring touch points to the datum table 0.D.:

- Datum plane, position finding
- Workpiece center = datum
- Circle center = datum

Error messages

If under parameter Q80 a datum number that does not exist in the datum table is addressed with the M functions M38, M39, the following error message is displayed:

DATUM NOT DEFINED

If you attempt to use M39 to transfer values to the datum table 0.D although the table is protected from editing and erasure (see chapter "File Management") the control will display the error message:

PROTECTED PGM!

TOUCH PROBE

Setup
Touch points in Cycle 3: Tool definition

In the "Manual" and "Electronic handwheel" modes of operation, the measured values from the probing functions described below can be transmitted to the "tool definition" cycle.

Prerequisite

Before starting the probe functions you must define and call the tool with the "tool definition" cycle in the "Program run/single block" or "Program run/full sequence" mode of operation (see chapter "Programming Modes", section "Coordinate Transformations, Cycle 3: Tool Definition" and "Tool call").

Example:

```
0 BEGIN   PGM 10   MM
1 CYCL DEF 3.0   TOOL DEF
2 CYCL DEF 3.1   T 1
                 R+0
3 CYCL DEF 3.2   X+0   Y+0
4 CYCL DEF 3.3   Z+0   C+0
5 TOOL CALL 1    Z   U+0
6 END PGM 10   MM
```

TOUCH PROBE

After you have defined and called the electrode you can execute the probe function.

Probe function

The following probe functions provide a menu item for transferring touch points to Cycle 3: „Tool definition":

- Datum plane, position finding
- Workpiece center = datum
- Circle center = datum

Error message

If you attempt to probe without having defined the tool in Cycle 3 or without a tool call, the control displays the error message:

TOOL DEF MISSING

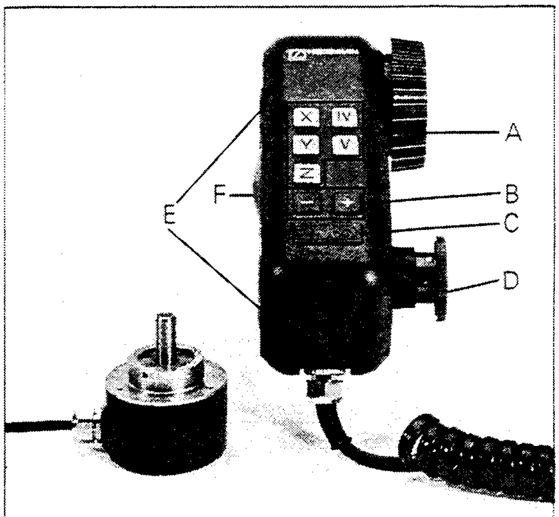
Versions

The control is usually equipped with an electronic handwheel. It can be used, for example, to set up the machine.

There are two versions of the electronic handwheel:

HR 130: to be incorporated into machine operating panel

HR 330: portable version with
axis selection keys (A),
axis direction keys (B),
rapid traverse key (C),
EMERGENCY STOP button (D).



HR 130 **HR 330**

Interpolation factor

The displacement per handwheel turn is determined by the interpolation factor (see table to the right).

Interpolation factor	Displacement in mm per turn
0	20.0
1	10.0
2	5.0
3	2.5
4	1.25
5	0.625
6	0.313
7	0.156
8	0.078
9	0.039
10	0.020

Operating the HR 130

The handwheel is switched to the required machine axis with the axis keys of the control.

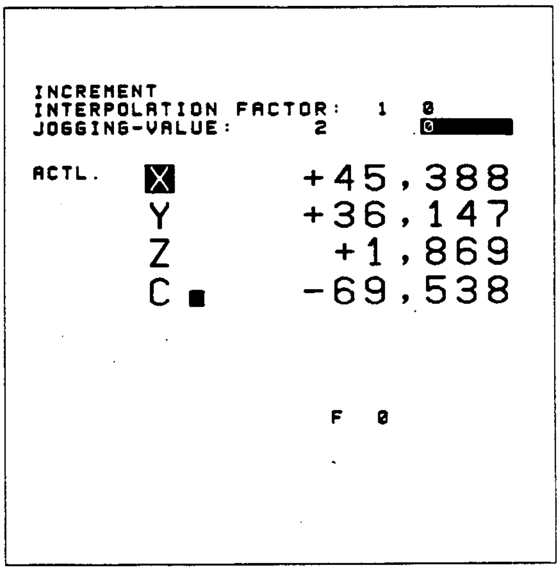
Operating the HR 330

The axis is selected on the handwheel.
The axis to be driven by the electronic handwheel is highlighted in the screen display.

The HR 330 is to be attached by its magnetic holding pads (E) to the machine tool such that unintentional activation is not possible.

When pulling the handwheel from the machine be careful not to press the axis direction keys (B).

If you wish to hold the handwheel unit during use, press the enabling switch (F) on the back of the housing. The handwheel direction keys will not function unless the enabling switch is activated!



In the "Electronic handwheel" operating mode, the machine axes can also be driven with the external axis direction buttons.

Operating the HR 130/330

Set operating mode and initiate the dialog

INTERPOLATION FACTOR: 3	Select the "Interpolation factor" dialog. Enter the desired interpolation factor, e.g. 4. Confirm entry.

INTERPOLATION FACTOR: 4	Y Select the axis: on the control (HR 130) or on the handwheel (HR 330)

The tool can now be moved in a positive or negative Y direction with the electronic handwheel.

Jog positioning

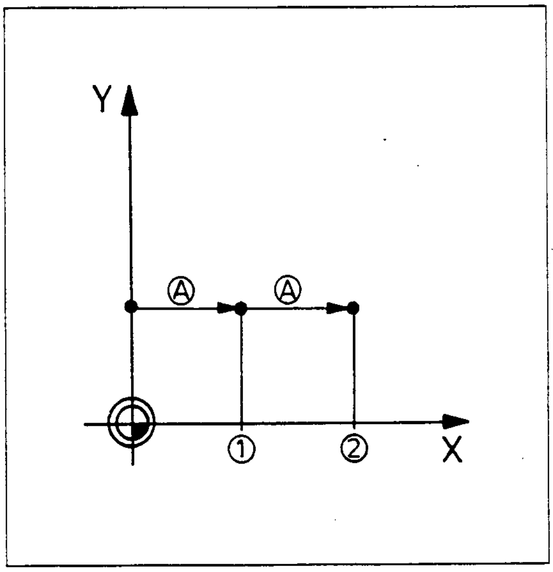
The machine manufacturer can activate incremental jog via the integral PLC. In this case, a traversing increment can be entered in this operating mode.

The axis is moved by the entered increment when you press an external axis button. This can be repeated as often as desired. Only single-axis movements are possible.

Ⓐ Jog increment: e. g. 2 mm.

① Axis direction button (e. g. X) pressed once.

② Axis direction button pressed twice.



Entering the jog increment

Set operating mode and initiate the dialog

JOG-INCREMENT: 1	Select "Jog-increment" dialog. Enter the jog increment, e. g. 2 mm. Confirm entry.

JOG-INCREMENT: 2	X or another axis button. The axis is driven by the entered jog increment.

Tool call/C axis/Rotational speed of C axis

You must first define (i.e. enter the dimensions of) a tool before you can call it with "TOOL CALL" in the "Positioning with MDI" operating mode. A tool is defined via "TOOL DEF" in the part program.

The concepts "TOOL DEF" and "TOOL CALL" are defined in the chapter "Programming Modes" under "Tool definition".

Initiate dialog — TOOL CALL

Example: Tool call

TOOL NUMBER ?	Enter tool number. ENT Confirm entry.

Select the tool axis

WORKING TOOL AXIS X/Y/Z/IV ?	Z Enter tool axis, e.g. Z.

Tool undersize

TOOL UNDERSIZE (DIAMETER)	Enter tool undersize. ENT Confirm entry.

Following electrode

FOLW. ELECTRODE YES=ENT/NO=NOENT	NO ENT If no, press NOENT.

BLOCK COMPLETE	START Start tool call.

Positioning with Manual Data Input
Positioning to entered coordinates

In the operating mode "Positioning with manual data input", single-axis positioning blocks can be entered and executed (the entered positioning blocks are not stored).

Traversing to position

Initiate the dialog — X or another axis key.

POSITION VALUE ?	I Incremental – absolute? ENT Enter a numerical value for the selected axis. Confirm the entry.

Radius compensation

TOOL RADIUS COMP.: R+/R–/NO COMP. ?	ENT Enter either no radius compensation or R+ R– enter desired radius compensation.

FEED RATE ? F = / FMAX = ENT	ENT Enter either the feed rate or ENT no value for rapid traverse, or NO ENT for feed rate from machine parameter.

MISCELLANEOUS FUNCTION M ?	ENT Either enter a miscellaneous function, e.g. M03 or NO ENT choose no miscellaneous function.

BLOCK COMPLETE	START Start the positioning block.

Terminate block entry

END

Direct termination of input. Data entered previously such as radius compensation, feed rate, or direction of C-axis rotation then remain permanently effective.

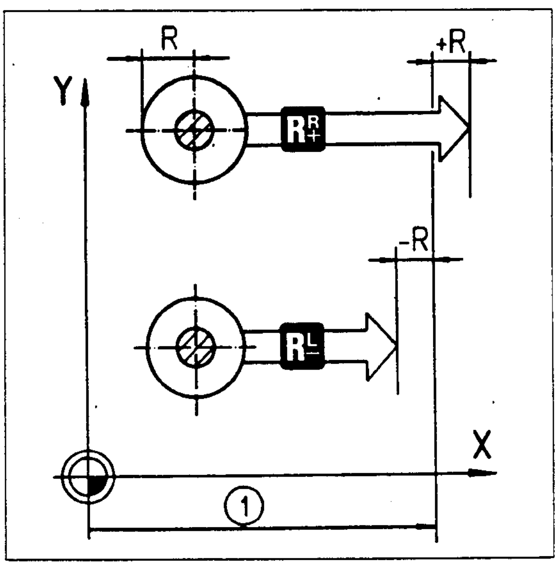
Single-axis radius compensation

For single-axis positioning blocks, you only have to consider whether the tool path is lengthened or shortened by the tool.

R+ tool path to be increased.
R– tool path to be reduced.

If a radius compensation R+/R– is also entered to position the tool axis, this axis is not compensated.

① Nominal position



Program Run
Single block, Full sequence

Stored programs are executed in the operating modes "Program run single block" and "Program run full sequence".

The workpiece datum must be set before machining the work!
See: Datum setting with/without probe system.

Program run single block

In this operating mode, the control executes the part program block by block. The program must be restarted after every block.

Program run single block is best used for program test and for the first program run.

Operating mode		Single block

Selecting the program

	PGM NR	Select the program or, if the program was already selected:
	GOTO 0 ENT	select block 0.

0 BEGIN PGM 7225	The first program block is shown in the current line of the program.

Starting the run

START	Each program block must be started with the machine START button.

Program run full sequence

In this operating mode, the control executes the machining program until a programmed stop or end of program occurs.

Stop functions:
M02, M30, M00 (M06 "STOP", if assigned a stop function via machine parameter).

The program run is also stopped if an error message appears.

You must restart the program to continue after a programmed stop.

Selecting the program

Operating mode — Full sequence
Select the program and block number as described above.

Starting the run

START	The program runs continuously until a programmed stop or end of program occurs.

Feed rate

The programmed feed rate can be varied via the feed rate override.

Q parameters

You can check and, if necessary, change Q parameters during program run or after interrupting program run.

Check* parameter

Q ☐ ENT Select and check the desired parameter.

Change parameter

ENT Terminate Q parameter display or

☐ ENT change the parameter and confirm.

If you wish to interrupt program run before checking the Q parameters you must first enter the following data.

Interrupt program run

STOP Stop program run by pressing the machine STOP button.

STOP Interrupt program run.

* Q parameters can be changed during a program run for erosion **without** erosion parameter tables. A program run must be interrupted to change Q parameters for erosion **with** parameter tables.

Program Run
Background programming

Programming during program execution

While a part program is being executed in the "Program run full sequence" operating mode, **another** program can, in the "Programming and editing" mode, be **simultaneously** either edited or transferred via the data interface RS-232-C/V.24.

This parallel operation is especially advantageous for long programs with little operator activity.

A program cannot be run and edited at the same time.

Starting the part program

Operating mode

Initiate the dialog

PROGRAM NUMBER =	ENT	Select part program.

START Start machining.

Parallel operating mode: programming and editing

Operating mode

PGM NR Select and edit the program

or

EXT transfer a program via the RS-232-C/V.24 data interface.

Screen display

The screen is divided into two halves during parallel operation: The program to be edited is shown in the upper half. The program currently in process appears in the lower half: program number, current block number and current status are displayed.

Terminating the parallel operating mode

Operating mode

Parallel operating is terminated by pressing the "Program run/full sequence" key.

Cycle STOP

The cycle STOP button is built into the machine control panel by the machine tool builder. This button interrupts a running erosion program and retracts the electrode by a maximum of 20 program positions back to the beginning of erosion.

This feature ensures that the electrode returns to its starting position along the same path which led to the point of interruption, thereby preventing damage to the workpiece or the electrode.

Example

⋮	
10 L X+50 Y+50 Z+2	Positioning to start erosion
11 L Z–5 F100 M36	First erosion step: sinking
12 L C+10	Second erosion step: rotating the C axis
13 L IX+15	Third erosion step: eroding in the X direction
14 L IZ–8	Fourth erosion step: sinking
⋮	

If you press the cycle STOP button during block 14, the control will return the electrode to block 10 by retracing its path back through blocks 13, 12, and 11.

Re-approaching the Contour

An interrupted erosion program can be restarted in the following manner:
In the "TEST RUN" mode, select a program number and confirm with ENT.

Run a program test up to the part program block number before the last executed step. The control calculates up to this point and restores the previous status (datum shift, radius compensation etc.).

If the axis display is set to NOML, the nominal position is shown in the status display. You can continue blockwise by pressing the key. The M-functions M3, M4, M5, M36 and M37 are automatically activated. If you wish other M-functions to be active, you must activate them beforehand in manual mode.

After switching to the "Program run/full sequence" mode, you can resume the program by pressing START. The electrode moves to the nominal positions in a fixed sequence: first the tool axis, then the C axis, then the X and Y axes.

Programming Modes (P)

Programming Modes (P)

Programming Modes (P)

Conversational Programming
General information

Introduction

The individual work steps on a conventional electrical discharge machine must be initiated by the operator. On an NC machine, the numerical control assumes computation of the tool path, coordination of the feed movements on the machine slides and generally also monitors the rotational speed of the C axis. The control receives the information for this in form of a part program in which the machining of the workpiece is described.

Program start and
specification of blank (for test graphics)

Define an erosion parameter table,
set the highest and lowest power stage

Define and call an electrode,
move to the electrode change position

Move to the workpiece contour,
generator ON,
machine the workpiece contour,
generator OFF,
depart from the workpiece contour

Traverse to the electrode position

End of program

Program scheme

Files

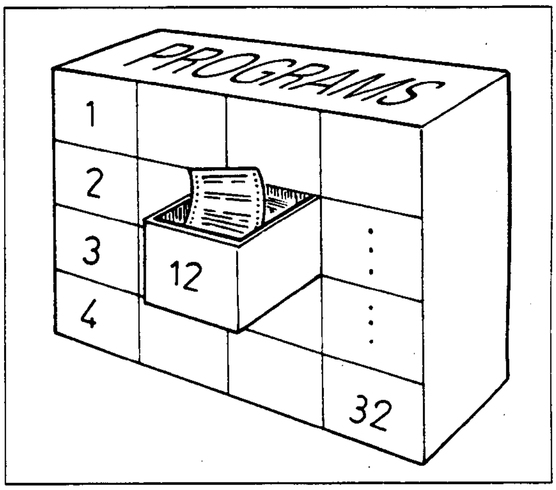
The control can manage up to 32 files (NC programs, erosion parameter tables, datum tables) in approx. 88K byte RAM memory.

The part program examples on the following pages will illustrate the TNC's conversational programming.

One part program can contain up to 1000 blocks.

The files are identified by their program numbers.

A file consists of individual lines.



Blocks

Every block in a program corresponds to one work step, e.g.
L X+20 Y+30 Z+50 R0 F1000 M36.

Block numbers (Sequence numbers)

The block number (also called the sequence number) identifies the program block in a part program. The control assigns a unique number to each block.

Words

Each block is composed of words, e.g. X+20;

Address Values

A word is composed of an address letter, e.g. X and a value, e.g. +20.

Abbreviations used above:

L = linear interpolation
X, Y, Z = coordinates
R0 = no tool radius compensation
F = feed rate
M = miscellaneous function

7	L	Z–20	R0 FMAX	M36
8	L	X–12	Y+60 R0 FMAX	
9	L	X+20	Y+60 RR F40	
10	RND	R+5	F20	
11	L	X+50	Y+20 RR F40	
12	CC	X–10	Y+80	
13	C	X+70	Y+51,715 DR+ RR	

The dialog principle

Program input is dialog guided, i. e. the control requests the required data. The corresponding dialog sequence for each program block is started with a dialog initiation key, e. g. "TOOL DEF" (the control subsequently requests the tool number, then the tool length, etc.).

Errors are displayed in plain language during program input. False entries can be corrected immediately – while entering the program.

Initiate dialog:	TOOL DEF Example: tool definition
First dialog query appears	Enter value ENT Transfer and continue dialog
Second dialog query appears	Enter value ENT Transfer and continue dialog
	END □ Terminate entry for current block

Responding to dialog queries/ Continuing the dialog

ENT

After you press a dialog initiation key, the control requests the necessary data. You must give a response to every dialog query. The response is shown in the highlighted field on the screen. After answering the dialog query, the entry is transferred to the memory with the "ENT" key. "ENT" is short for "enter" (i. e. confirm entry, transfer, store). The control then displays the next dialog query.

Skipping dialog queries

NO ENT

To make the entries in the preceding block modal, that means valid for the current block, (e. g. feed rate or spindle speed), do not respond to the associated dialog queries; skip them with the "NO ENT" key.

Entries already displayed in the highlighted field or already included in the program are deleted with "NO ENT"; the next dialog query appears on the screen.

During program run, the previously programmed values are valid for the associated address.

```
PROGR. AND EDITING            1
FEED RATE ?         F= / F MAX = ENT
2    TOOL CALL 1              Z
          U+0,1
3         Z+10
                     R0 F        M36
4         Z+15
                     R0 F        M03
5         Z+20
                     R0 F        M08
------------------------------------
ACTL. X       +45,388   Y    +36,147
      Z        +1,869   C    -69,538

                       F
```

Directly terminating a block

END □

If you have programmed all the desired information in a block, you can directly terminate the block with "END □".

The control saves the entered data, and no more queries for this block appear.
Data not programmed in this block remain effective as programmed in previous blocks.
Certain routines, such as "Read-in program", are also terminated with this key.

Entering numerical values

Numerical values are entered with the numeric keypad – with a decimal point or decimal comma (selectable via machine parameter) and sign key. You need not enter preceding or succeeding zeroes. You can enter the sign before, during or after the entering the number.

Conversational Programming
Editing functions

Editing

The term **editing** means entering, changing, supplementing and checking programs.

The editing functions are helpful in selecting and changing program blocks and words, and they become effective at the touch of a key.

Selecting a block

GOTO

The current block stands between two horizontal lines.

A specific block is selected with "GOTO □".

Initiate the dialog — GOTO □

GOTO: NUMBER =	ENT	Key in and confirm the block number.

Paging through the program

Vertical cursor keys:

Select the next lower or next higher block number.

Hold down a vertical cursor key to continuously run through the program lines.

Inserting a block

You can insert new blocks anywhere in existing programs. Just call the block which is to precede the new block. The block numbers of the subsequent blocks are automatically increased.

If the program storage capacity is exceeded, this is reported at dialog initiation with the error message: = PROGRAM MEMORY EXCEEDED =.

This error message also appears if program end (PGM END block) is selected. You should then select a lower block number.

Editing words

Horizontal cursor keys:

The highlighted field is moved within the current block and can be placed on the program word to be changed.

One word in the current program block is to be changed:	→	Move the highlighted field to the word to be changed.

The dialog query appears for the highlighted word, e. g.		
COORDINATES ?	X	
		Change the value.

To change another word:	← →	Move the highlighted field to the word to be changed.
If all corrections have been made:	END □	Transfer the block (or move the highlighted field to the right or left off the screen).

Searching for lines with certain addresses

You can use the vertical cursor keys to search for lines containing a certain address in the file.

Use the horizontal cursor keys to place the highlighted field on a word having the search address, and then page in the file with the vertical cursor keys:

Only those blocks having the desired address are displayed.

Example ME

All lines with the address M are to be displayed:	↓ ↑	Select one block with the desired address.
	→	Place the highlighted field on a word with the required address.
MISCELLANEOUS FUNCTION M ?	↓ ↑	Call blocks with the desired address.

Delete block

The current line (in a file) is deleted with **DEL □.**
Select the line you wish to delete with **GOTO** □ or a cursor key.
Program lines can only be deleted in the PROGRAMMING AND EDITING operating mode.
After deletion, the line with the next lower number appears in the current line.
The following numbers are corrected automatically.

The current line is to be deleted:	Delete block.

Delete file section

To delete file sections, call the last line of the section that you wish to delete.
Then continue pressing **DEL** □ until all blocks in the definition or program section are deleted.

Clear entry, error message

You can clear numerical inputs with the "CE" key. A zero appears in the highlighted field after pressing the "CE" key.
Non-blinking **error messages can also be cleared with the "CE" key.**

An entered value and the address are completely cleared with "NO ENT".

File Management
Creating a file
Selecting an existing file

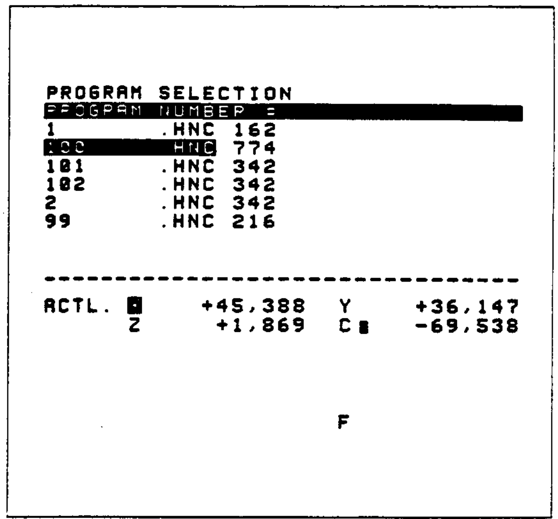
You create files and select stored files by first pressing the "PGM NR" key (program number).

A table with the files (NC programs, erosion parameter tables and datum tables) stored in the TNC appears on the screen. The program number last selected is highlighted. The program length in characters is given after the program number.

You can select the desired file either
- via the cursor keys

or
- by entering its number.

If the selected file number does not yet exist, a new file is created.



Creating a file

To create new files (NC programs, erosion parameter tables, datum tables) proceed as follows:

Initiate the dialog — PGM NR

PROGRAM SELECTION	↓ ↑ ENT	Move the highlight to "program selection".
PROGRAM NUMBER =	ENT	Enter the program number (the datum table has the number 0) and confirm entry.

ENT = HNC / NO ENT	ENT	for programs in HEIDENHAIN conversational dialog.
MM = ENT / INCH = NO ENT	ENT	for dimensions in mm, or
	NO ENT	for dimensions in inch.

ENT = ERODING TABLE / NO ENT	NO ENT ENT	for erosion parameter tables.

Example display

0 BEGIN PGM 96231 MM
1 END PGM 96231 MM

Selecting an existing program

All existing files can be edited and NC programs tested, displayed graphically and executed, regardless of the selected type of programming.

Initiate the dialog — PGM NR

PROGRAM SELECTION		
PROGRAM NUMBER =	↓ ↑ ENT	Place the highlighted field on the desired program number.
or	ENT	Enter the program number.

Example display

0 BEGIN PGM 7645 MM
1 BLK FORM Z X+0
Y+0 Z–40
2 BLK FORM X+100
Y+100 Z+0

PGM NR

File Management
Program protection/erasure
Program directory (EPROM)

Program protection

After creating a program, you can designate it as erase- and edit-protected.
The file is then marked with a **P** ("protected") in the file directory and at the start and end of the program.
Protected files can be executed and viewed, but not changed.
A protected program can only be erased or changed if the erase/edit protection is removed beforehand.

Activating edit protection

Initiate the dialog — PGM NR

PROGRAM PROTECTION	↓ ↑ ENT Move the highlight to "program protection" and confirm selection.
	↓ ↑ Move the highlight to the desired program.
YES = ENT / NO = NO ENT	ENT Activate program protection.
	NO ENT Cancel program protection.
	END □ Return to the file management functions.

Erasing a program

Files that are not protected can be erased.

Initiate the dialog — PGM NR

ERASE PROGRAM	↓ ↑ ENT Move the highlight to "erase program" and confirm selection.
	↓ ↑ Move the highlight to the desired program.
ERASE = ENT	ENT Erase the file.
	END □ Return to the file management functions.

Program directory (EPROM)

Erosion parameter tables placed by the machine tool builder in the EPROM can be shown in a directory.
Like erosion parameter tables in the RAM, these files can be defined in the "generator" cycle.

Program Selection
Blank form definition

Test graphics

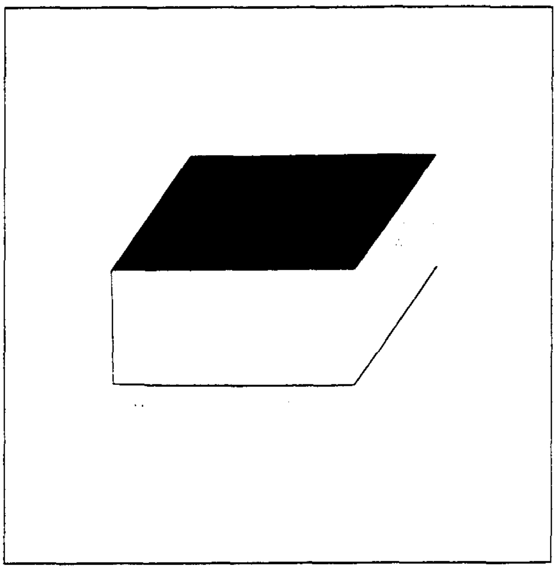
A blank form definition must be programmed before the machining program can be simulated *graphically.*

Blank

For the graphic displays, the blank dimensions of the workpiece must be entered at the start of program.

The blank form must always be programmed as a cuboid, aligned with the machine axes.

Maximum dimensions:
14 000 mm x 14 000 mm x 14 000 mm.

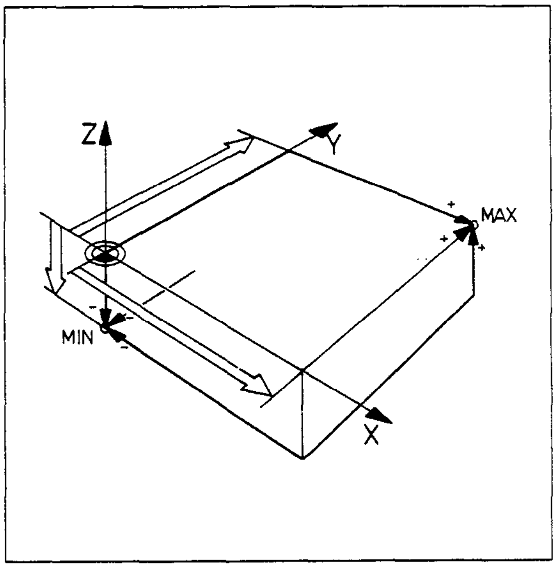


Minimum point
Maximum point

The cuboid is defined with the minimum point **(MIN)** and maximum point **(MAX)** (points with "minimum" and "maximum" coordinates).

MIN can only be entered in absolute dimensions; MAX may also be incremental.

The blank data are stored in the associated machining program and are available after program call.



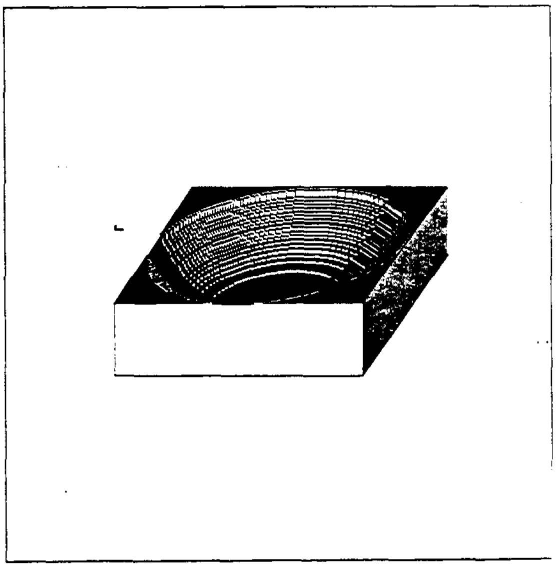
Graphic display

Machining can be simulated in the three main axes – with a fixed tool axis.

Tool form

Machining is correctly displayed with a cylindrical tool in the graphic view.

The graphic must be interpreted accordingly when using form tools.



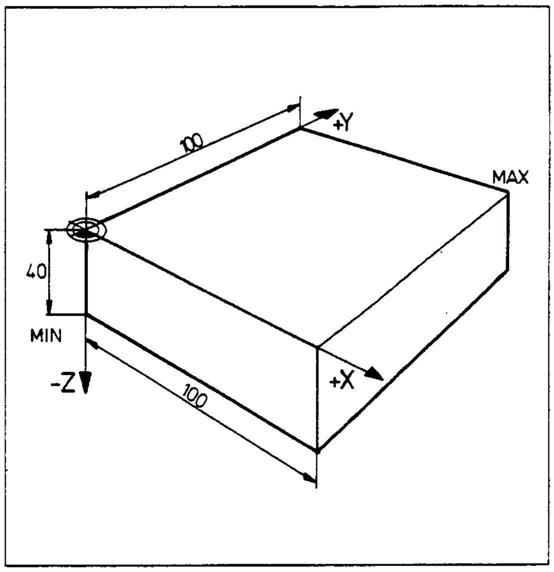
Program Selection
Blank form definition

BLK FORM

Example

The **MIN** point has the coordinates
X0, Y0 and Z–40.

The **MAX** point has the coordinates
X100, Y100 and Z0.



To define a blank, a program must be selected in the "Programming and editing" operating mode.

Entering the cuboid corner points

Initiate the dialog BLK FORM

TOOL AXIS PARALLEL X/Y/Z/4 ?	Z	Enter the electrode axis, e. g. Z.

MIN

DEF BLK FORM: MIN-CORNER ?	0 ENT	X coordinate.
	0 ENT	Y coordinate.
	+/- 4 0 ENT	Z coordinate.

MAX

DEF BLK FORM: MAX-CORNER ?	1 0 0 ENT	X coordinate.
	1 0 0 ENT	Y coordinate.
	0 ENT	Z coordinate.

Example display

1 BLK FORM 0.1 Z X+0
Y+0 Z–40

2 BLK FORM 0.2 X+100
Y+100 Z+0

Error messages

BLK FORM DEFINITION INCORRECT
The MIN and MAX points are incorrectly defined, or the machining program contains more than one blank definition, or the side proportions differ too greatly.

PGM SECTION CANNOT BE SHOWN
Wrong tool axis is programmed.

Tool definition in part program

Tool definition

TOOL DEF

If you wish to erode a programmed contour – consisting of straight lines and circular arcs – taking tool length and tool radius into account (contouring erosion with radius compensation) then you **must** enter both length and radius in the tool definition (TOOL DEF).

These data are programmed in the tool definition.

These tool definitions can be defined either individually in each part program or centrally in an arbitrarily named tool program that can be called through a PGM CALL.

Tool number

Compensation values always refer to a certain tool designated by a number.

Valid tool numbers: 1 to 99999999.

Tool definition in the part program

If tools required in a program are defined in that program, a program printout will include the specifications of the tool dimensions.

Input

Initiate the dialog TOOL DEF

TOOL NUMBER ?	ENT Enter the tool number. Tool number 0 cannot be programmed under TOOL DEF. Tool 0 is internally defined with L = 0 and R = 0.

TOOL LENGTH L ?	+/– ENT Enter the tool length or the difference to the zero tool.

TOOL RADIUS R ?	ENT Enter the tool radius.

Tool definition through cycle 3

The tool definition can also be programmed as standard cycle 3 with tool radius compensation in up to 4 axes (see "Standard Cycles").

Tool Definition
Tool length L

Tool length L

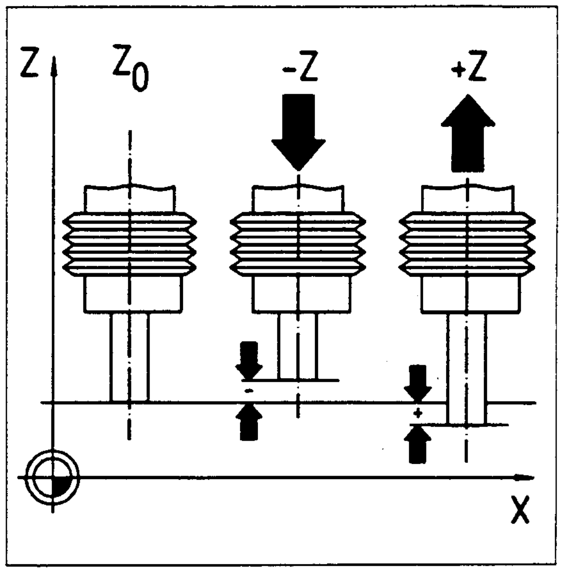
The tool length is compensated with a single adjustment of the tool axis by the length compensation.

Compensation becomes effective after tool call and subsequent movement of the tool axis.

Compensation ends after a tool is called or with T_0 (tool with a length of 0).

The correct compensation value for the tool length can be determined on a tool presetter or on the machine.

If the compensation value is to be determined on the machine, then you must first enter the workpiece datum.



Length differences

When the compensation values are determined on the machine, the zero tool serves as a reference.

The length differences –Z or +Z of the other clamped tools to this zero tool are programmed as tool length compensations.

If a tool is **shorter** than the zero tool, the difference is entered as a **negative** tool length compensation. If a tool is **longer** than the zero tool, the difference is entered as a **positive** tool length compensation.

Preset tools

If a tool presetter is used, all tool lengths are already known. The effective compensation values correspond to the tool length and are entered with the correct signs according to a list.

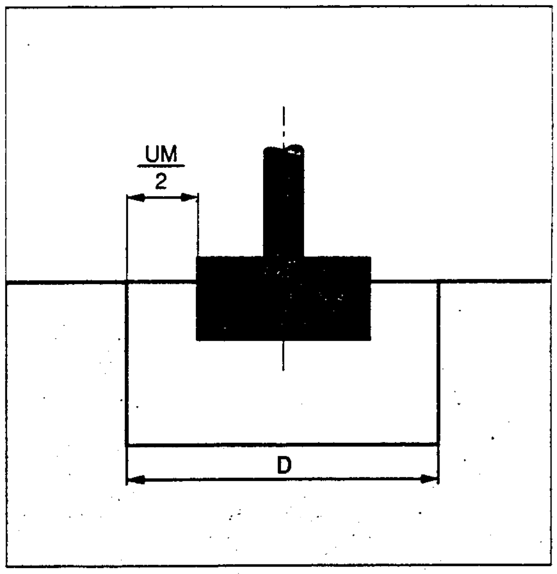
Tool Definition
Tool radius R

Sinking erosion

For sinking erosion, the actual electrode radius Re equals the tool radius R to be programmed in TOOL DEF.

The required cavity diameter D and the undersize UM (equals minimum undersize UNS) given in the erosion parameter table result in a radius:

$$R = Re = \frac{D - UM}{2}$$

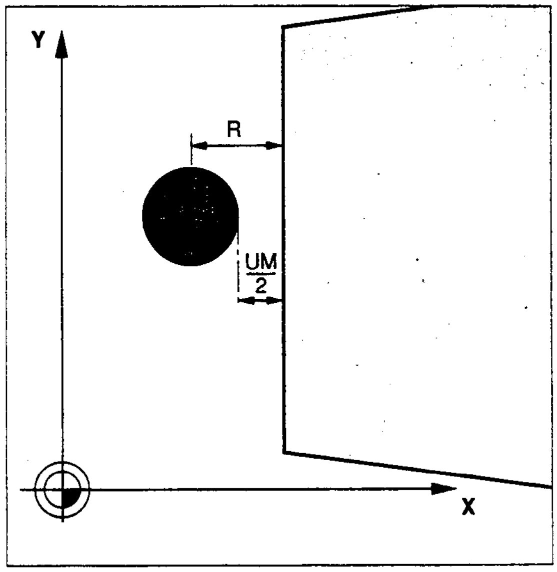


Contouring erosion with radius compensation

For contouring erosion with radius compensation the actual electrode radius Re can be chosen according to the required contour.

The tool radius R programmed in TOOL DEF results from the actual electrode radius Re and the undersize UM (equals minimum undersize UNS) given in the erosion parameter table.

$$R = Re + \frac{UM}{2}$$



Erosion with the disk cycle

In the disk cycle the actual electrode radius Re equals the tool radius R to be programmed in TOOL DEF.

Any actual electrode radius Re can be chosen, provided that it is greater than the expansion radius RAD to be programmed in the disk cycle.

Because of the relationship

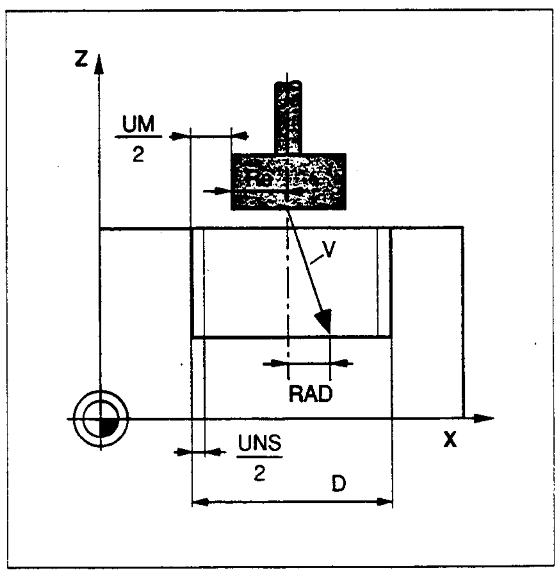
$$\frac{UM}{2} = \frac{UNS}{2} + RAD$$

the required disk diameter D, the minimum undersize UNS given in the erosion parameter table and the expansion radius RAD result in a radius:

$$R = Re = \frac{D - UM}{2}$$

or

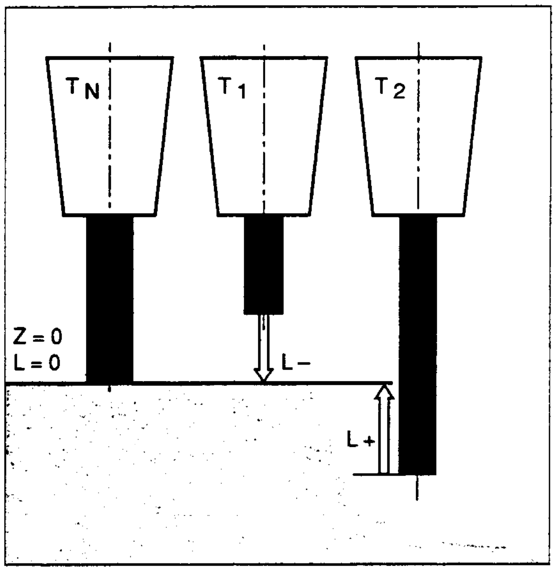
$$R = Re = \frac{D - UNS}{2} - RAD$$



Tool Definition
Transferring tool length

Tool lengths can be easily and quickly entered with the teach-in function.

1. Move the zero tool to the work surface and set the tool axis to zero.
2. After exchanging, move the tools T_1 or T_2 to the work surface.
3. Transfer each tool-axis display value in this position to the tool length definition. This gives you the length compensation to the zero tool.

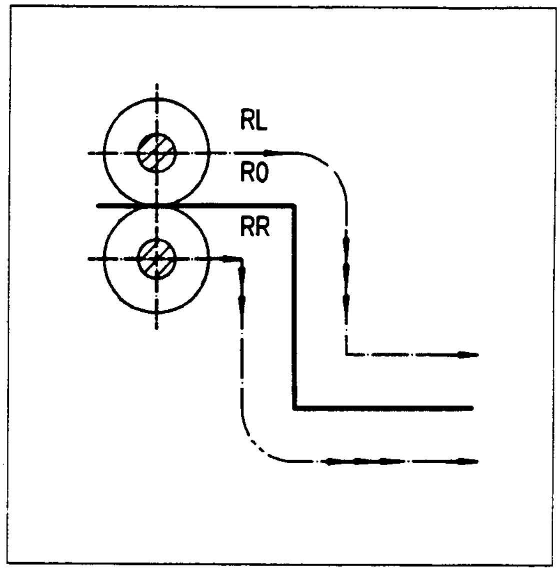


Input

Operating mode	or
	X Y Z Touch the surface with the zero tool.
Initiate the dialog	Z Spindle axis, e. g. Z.
DATUM SET	ENT Reset to zero.
	X Y Z Also touch the surface with the new tools T_1 or T_2.
Operating mode	
	Call a tool definition in a program and initiate the dialog "TOOL LENGTH L ?".
TOOL LENGTH L ?	Z Select the tool axis to transfer the tool length.
	ENT Transfer the length compensation.
TOOL RADIUS R ?	ENT Enter the radius.

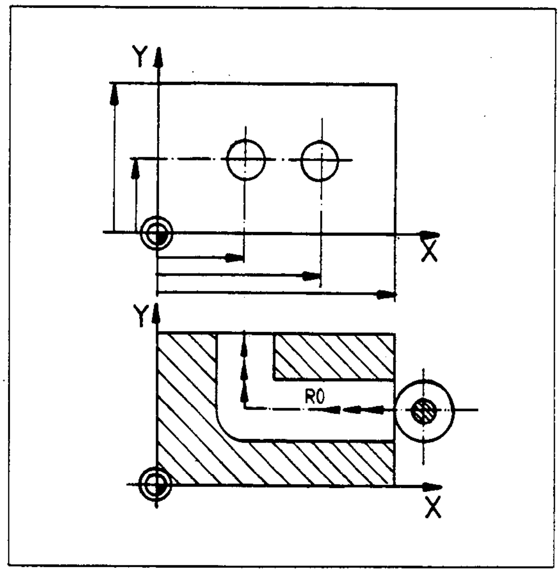
Electrode Path Compensation
Entering RL/RR

To automatically compensate for the tool radius – as entered in the TOOL DEF blocks – the control must be informed whether the electrode is moving to the left of, to the right of, or directly on the programmed contour.



R0

If the electrode is to travel **on** the programmed contour, no radius compensation should be programmed in the positioning block.
At the dialog query
TOOL RADIUS COMP.: RL/RR/NO COMP. ?
press the "ENT" key.
Screen display: R0



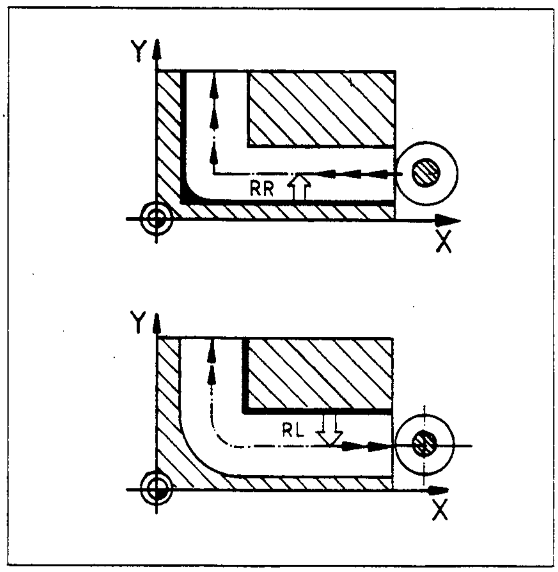
Programming radius compensation

The radius compensation is entered in positioning blocks (L, C etc.) with the "RL" and "RR" keys at the dialog query
TOOL RADIUS COMP.: RL/RR/NO COMP. ?

"Left" or "right" should be understood as looking in the direction of movement.

RR

If the electrode is to travel at the distance of the radius to the **right** of the programmed contour, press the "RR" key.
Display: RR



RL

If the electrode is to travel at the distance of the radius to the **left** of the programmed contour, press the "RL" key.
Display: RL

R

If the previous compensation should remain effective (modal):
press the "NO ENT" key.
Display: R

Electrode Path Compensation
Working with radius compensation

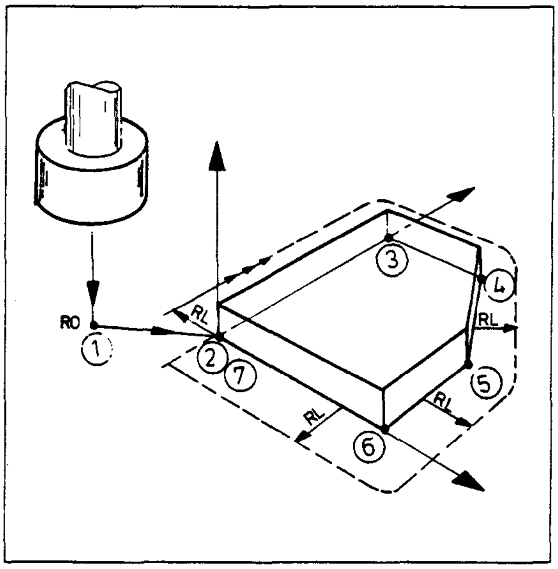
Starting point R0

Change the electrode and call the compensation values with "TOOL CALL".

Traverse rapidly to the starting point ①.

At the same time lower Z to the working depth (if danger of collision, first traverse in X/Y, then separately in Z!). This compensates for the electrode length.

The radius compensation still remains switched off with "R0".



1st contour point RL/RR

Traverse to contour point ② with radius compensation RL/RR at reduced feed rate.

Machining around the contour

Program the following contour points to ⑦ at erosion feed rate.

Since the RL/RR asignment remains constant, the associated dialog queries can be skipped with "NO ENT" or "END □".

Last contour point RL/RR

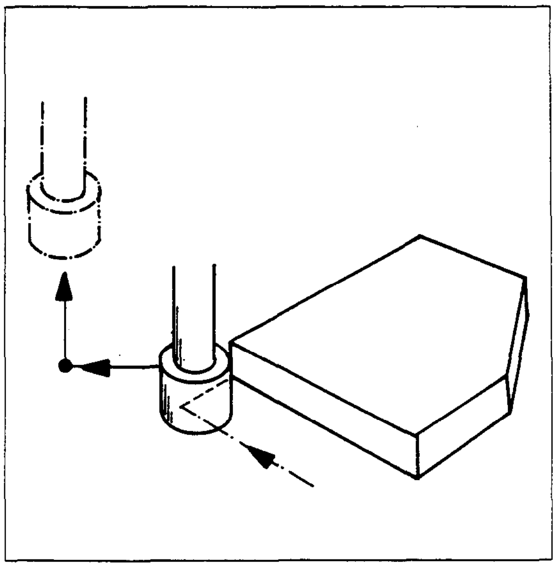
After a complete circulation, the last contour point ⑦ is identical to the first contour point ② and is still radius compensated.

End point R0

The end point (outside the contour) must be programmed without compensation R0 for complete machining.

To prevent collisions, only retract in the machining plane to cancel the radius compensation.

Then back-off the tool axis separately.



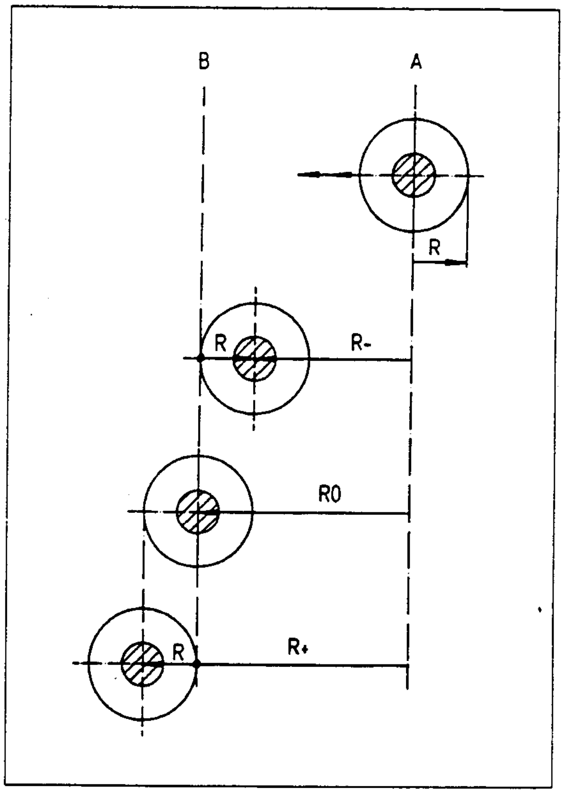
Electrode Path Compensation
Radius compensation R+, R–

Initiating the dialog
X Y ...

By pressing "R+" or "R–", you can lengthen or shorten a single-axis displacement by the electrode radius.

This simplifies:
- positioning with manual data input,
- single-axis machining.

The input dialog may be initiated directly via the corresponding yellow axis key.



Effect
RL RR

This radius compensation has the following effect:

- The displacement is **shortened** by the electrode radius: display **R-**.
- The electrode traverses **to** the programmed nominal position: display **R0**.
- The displacement is **lengthened** by the electrode radius: display **R+**.

R+/R– do not affect the tool axis.

Example

The electrode is to traverse from initial position X = 0 to X = (46 + electrode radius).

Initiate the dialog X

POSITION VALUE ?	4 6 ENT
TOOL RADIUS COMP.: R+/R–/NO COMP. ?	RR END Display: X+46 R+

Mixing
L and X

Uncompensated blocks (e. g. L X+20 R0) and single-axis blocks (e. g. X+20 R0 or X+20 R+) can be mixed in a part program.

Single-axis compensated positioning blocks (R+/R–) and radius compensated positioning blocks (RR/RL) are not to be entered in succession!

Correct:

```
L X+15 Y+20  R0
  Y+50       R0
  X+40       R+
  Y+70       R0
```

Incorrect:

```
L X+15 Y+20  RR
  Y+50       R+
L X+50 Y+57  RR
```

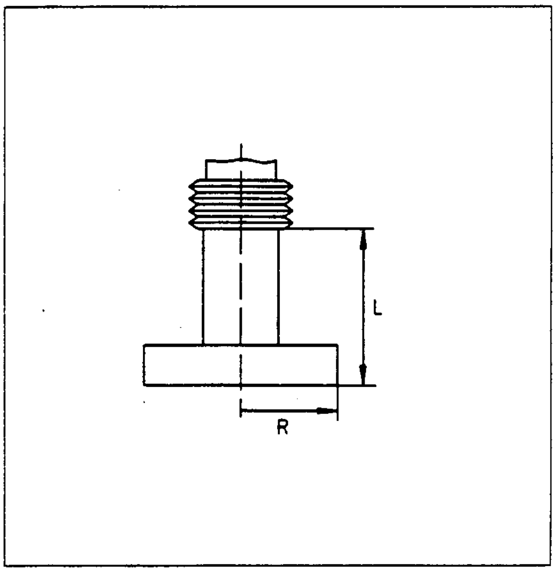
Tool Call

Tool call

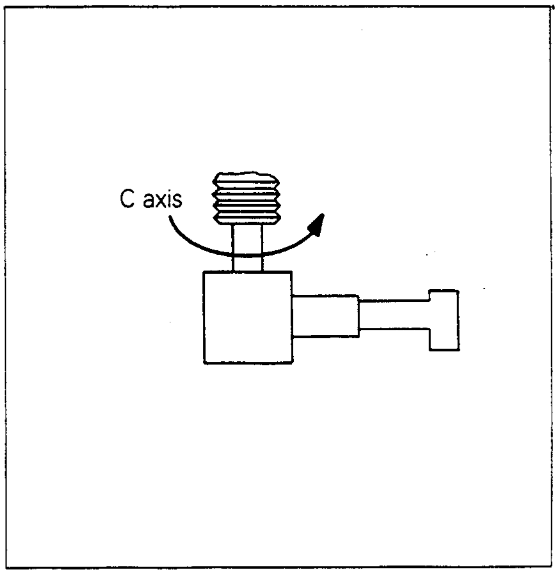
TOOL CALL

With **TOOL CALL** a new tool is called up. The values for length L and radius R given in **TOOL DEF** are compensated depending on the erosion process.

	Process		
	Sinking erosion	Disk cycle	Contour. erosion
Length comp.	●	●	●
Radius comp.	–	–	●



C axis as tool axis



Tool axis

After the tool number, the tool axis must be entered. It defines the working plane (the plane for circular movements, radius compensation and for the mirroring, rotation and scaling cycles).

Tool axis	Length compensation	Radius compensation
Z	**Z**	**XY**
Y	Y	ZX
X	X	YZ
C (IV)	no compensation	

Electrode undersize

The electrode undersize UM is entered immediately after the tool axis.

The actual electrode radius Re must always be smaller than the erosion radius by the amount of the erosion gap G and the maximum surface roughness.

The undersize value UM for **contouring erosion** and **sinking erosion** is determined
- above all, by the width of the erosion gap G
- but also by the maximum surface roughness Rmax (only for roughing!).

The undersize UM can be taken from the erosion parameter table. It equals the minimum undersize UNS.

For roughing:

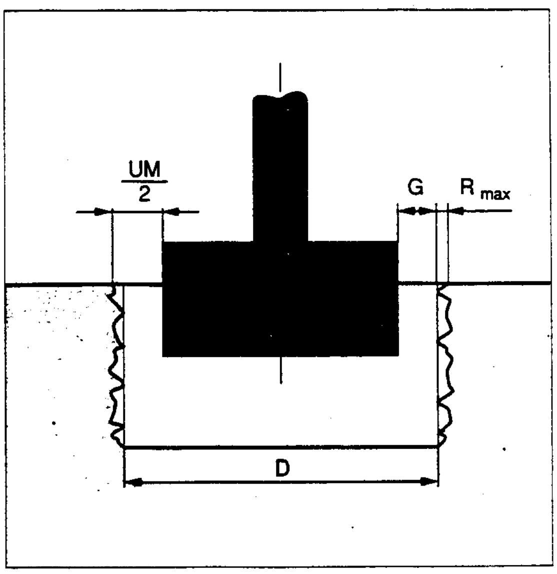
$$UM = 2G + 2 \cdot R_{max} = UNS$$

For finishing:

$$UM = 2G = UNS$$

The undersize value UM for **erosion with the disk cycle** is determined by
- the required disk diameter D
- and the electrode radius Re.

$$UM = D - 2\,Re$$



Tools
Tool call

Activating compensation

Tool length compensation becomes effective upon the next movement of the tool axis.
It can be seen as a single infeed height movement.
Tool radius compensation does not become effective until the compensation direction "RL" or "RR" is programmed in a positioning block.

Ending compensation

A "TOOL CALL" block ends the "old" tool length and tool radius compensation and calls the compensation values of the new tool.
Example: TOOL CALL 12 Z U+2

Tool radius compensation is also ended by programming "R0" in the positioning block.

If only the electrode undersize is entered with "TOOL CALL", the compensations remain valid.
Example: TOOL CALL U+2

Tool call

Initiate the dialog TOOL CALL

TOOL NUMBER ?	ENT	Enter the tool number.

Spindle axis

TOOL AXIS X/Y/Z/4 ?	Z	Enter the electrode axis, e.g. Z.

Tool undersize

TOOL UNDERSIZE (DIAMETER) ?	ENT	Enter undersize, e.g. 0.5.

Following electrode

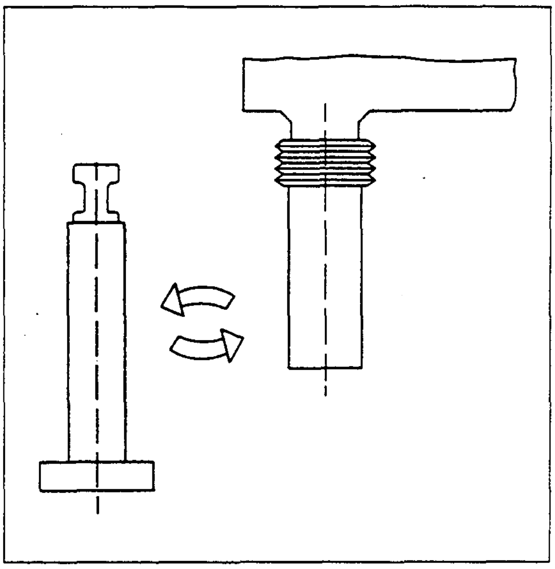
FOLLOWING ELECTRODE YES/NO ?	ENT	If no, press NO ENT.

Tool Call
Tool change

Tool change position

To change the tool, the electrode must be retracted in the tool axis and the C axis must be stopped.

We recommend the insertion of an additional block in which the axes of the machining plane are likewise backed-off.



Workpiece-related change position

The tool moves to a **workpiece-related** position if no additional measures are taken.

Example: L Z+100 FMAX M06
The tool is driven 100 mm over the work surface if the tool length is 0 or TOOL CALL 0 was programmed.

T0 reduces the distance to the workpiece (danger of collision!) if a positive length compensation was effective prior to TOOL CALL 0.

Machine-related change position

You can use M91, M92 or a PLC positioning to traverse to a **machine-related** tool change position.

Example: L Z+100 FMAX M92
(see Machine-related coordinates M91/M92).

Manual tool change

The program must be stopped for a manual tool change. Therefore, enter a program STOP before the TOOL CALL. M6 has this stop effect when the control is set accordingly via machine parameters. The program is then stopped until the external START button is pressed.

Automatic tool change

The tool is changed at a defined change position. The control must therefore move the tool to a machine-specific change position. The program run is not interrupted.

```
1 BLK FORM 0.1 Z X+0 Y+0 Z–40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL DEF 1 L+0 R+5
4 TOOL DEF 2 L–2.4 R+3
5 TOOL CALL 0 Z
6 L Z+200 R0 FMAX M06
7 TOOL CALL 1 Z U+1
8 L X+25 Y+30 FMAX
9 L Z+2 FMAX M3
.
.
.
```

Feed rate
F

The feed rate F, i. e. the traversing speed of the tool in its path, is programmed in positioning blocks in mm/min or 0.1 inch/min. The current feed rate is shown in the status display on the lower right of the screen.

If no feed rate is programmed, the feed rate is read from the general user parameter MP 1090. You do not need to enter a feed rate in the NC program!

Feed rate override

The feed rate can be varied within a range of 0% to 150% with the feed rate override on the control operating panel.

Rapid traverse

The maximum input value (rapid traverse) on the control for positioning is:
- 29 998 mm/min or
- 11 800/10 inch/min.

The maximum operating speeds are set for each axis.
FMAX or the max. input is programmed for rapid traverse.
The control automatically limits rapid traverse to the permissible values.

FMAX is only effective **blockwise.**

If the F display is highlighted and the axes do not move, this means the feed rate was not enabled at the control interface. In this case, you must contact your machine manufacturer.

Rotational speed of the C axis

The rotational speed of the C axis is set in the Q parameter (see "Parametric Programming, Special functions").

If the rotational speed is not programmed via Q parameter, or if the corresponding Q parameter is assigned the value 0, the speed will be taken from the general user parameter MP 2090. This rotational speed programming is only valid, however, for free rotation of the C axis with the aid of the miscellaneous function M.

If the C axis is moved by pressing the axis direction button, the rotational speed will be adjusted to the programmed feed rate.

Miscellaneous functions
M

Miscellaneous functions can be programed to regulate certain machine functions (e. g. rotation of C axis), to control program run and to influence tool movements. The miscellaneous functions are comprised of the address M and a code number according to ISO 6983. All of the M functions from M00 to M99 can be used.

Certain M functions become effective at the start of block (e. g. M36: Erosion "on"), i. e. before movement, and others become effective at the end of block (e. g. M09: flushing "off"). A list of all M functions with their effects as determined by the control can be found inside the back cover.

Only a certain number of these M functions are effective on a given machine.
Some machines may employ additional, non-standard M functions not defined by the control.
M functions are normally programmed in positioning blocks (L, C etc.).
However, M functions can also be programmed without positioning:

STOP

- Via the "STOP" key or
- by initiating the dialog with the "L" key and skipping the queries with "NO ENT" up to address M.

Erosion functions

M03: Free rotation of the C axis in clockwise direction (CW)*
M04: Free rotation of the C axis in counterclockwise direction (CCW)*
M05: C axis stop

M08: Flushing on
M09: Flushing off

M36: Erosion on (gap control active)
M37: Erosion off (gap control active)

* The direction of rotation is determined by the machine tool builder.

Programmable Stop/Dwell Time

Stopping program run

STOP

Program run can be stopped by one of the following functions.
Restart by pressing the external START button.

Initiate the dialog STOP

MISCELLANEOUS FUNCTIONS M ?	
Miscellaneous function is desired:	ENT Enter the miscellaneous function.
No miscellaneous function desired:	NO ENT No entry.

Example

18 STOP	M	Program run is stopped at block 18. No miscellaneous function.

M02/M30

- Program stop.
  Return to block 1 of the program.

M00

- Program stop.

M06

- Program stop and tool change.

  Program stops only when set accordingly by machine parameter!

Dwell time

Cycle 9 "Dwell time" can be used during program run to delay execution of the next block for the programmed time period (see "Other cycles").

Note:
The program continues running after the dwell time runs out!

Path Movements
Entry

The control/operator dialog for entering positioning blocks is illustrated below using the example of a straight line movement.

Operating mode	Programs can only be input in "PROGRAMMING AND EDITING".
Initiate the dialog	Select the type of movement, e.g. straight line.

Example

COORDINATES ?	Enter the end point of movement:
	X Select the axis, e.g. X.
	I Incremental – absolute ?
	Enter numbers with sign.
	Y Enter further coordinates.
	⋮
	ENT If all endpoint coordinates are entered, confirm the entry.

TOOL RADIUS COMP.: RL/RR/NO COMP. ?	RL RR Enter the radius compensation or
	ENT no radius compensation (R0).

FEED RATE ? F =	ENT Enter the feed rate or press only
	ENT for FMAX = rapid traverse.

MISCELLANEOUS FUNCTION M ?	ENT Enter a miscellaneous function if desired.

Abbreviated input

Subsequent blocks can be ended immediately with "END □", e.g. after entering the corner point coordinates.

In these cases, the last entries remain valid for non-programmed addresses.

Addresses may be skipped with "NO ENT".

Path Movements
Initiating the dialog

Contour elements

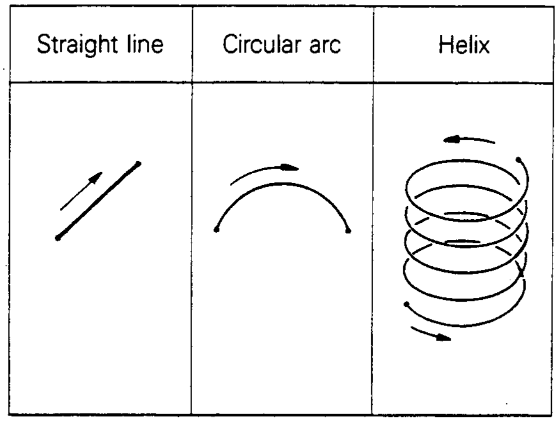
The shape of the workpiece is programmed without considering the tool. You always program as though the tool moves, regardless of the machine design. The programmable contours are composed of the contour elements **straight line** and **circle.** Using tool radius compensation, the control computes the tool-dependent path for the cutter center along which the tool is guided.

Generating the workpiece contour

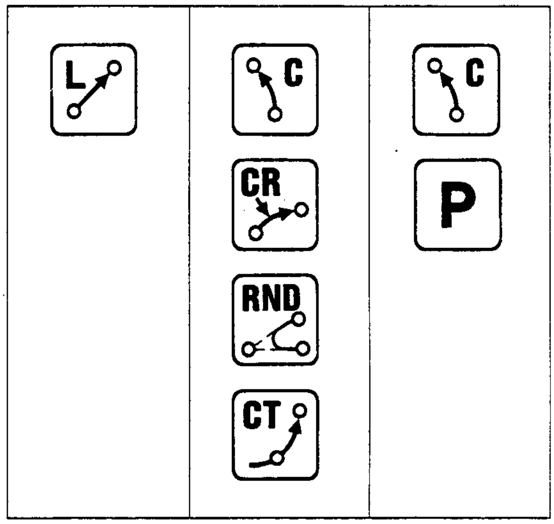
To be able to generate the workpiece contour, the control must be given the individual contour elements. Since each program block specifies the next step, the following information is required:

- straight line or circle
- the coordinates of each endpoint or other geometrical data such as the circle center and contour radius.

Contour elements



Path function keys



Initiating the dialog

To program a contour element, always begin with one of the gray path function keys. The type of movement is then defined for the contour element in question.

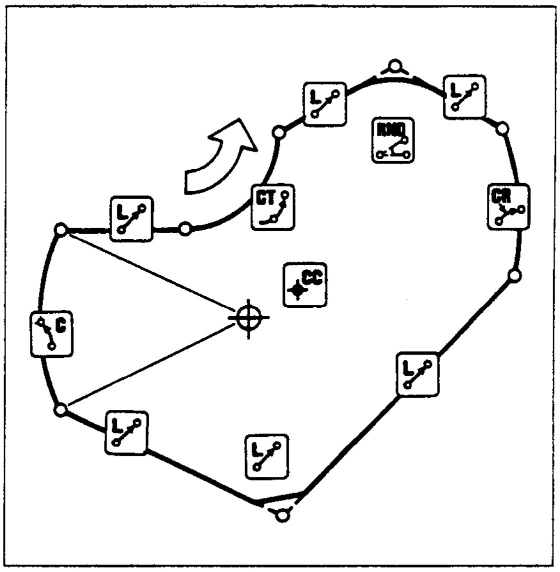
Coordinates

Point coordinates can only be input after selecting the path function.

Incremental/ Absolute

I

To enter the point coordinates incrementally, press the key for incremental inputs.



Straight lines

Straight line (L):
The electrode moves in a straight line.
The endpoint of the straight line must be programmed.

Chamfer
A chamfer is inserted **between two straight lines.**

Circles

Circle center (CC) –
also the pole for polar coordinates:
Used to program the circle center for a circular arc with the "C" key, or to program the pole for polar coordinates.
CC generates no movement!

Circular movement (C):
The electrode is moved in a circular arc. Program the endpoint of the arc. The circle center must be specified beforehand.

Corner rounding (RND):
An arc with tangential connections is inserted between two contour elements.
Program the arc radius and (in other blocks) the contour elements of the corner to be rounded.

Circular arc (CT) = "circle tangential":
A circular arc is tangentially connected to the preceding contour element.
Only the endpoint of the arc is programmed.

Circular arc (CR) = „circle per radius":
The tool is moved on a circular path.
Program the circle radius and the endpoint of the arc (but not the circle center).

Multi-axis movements

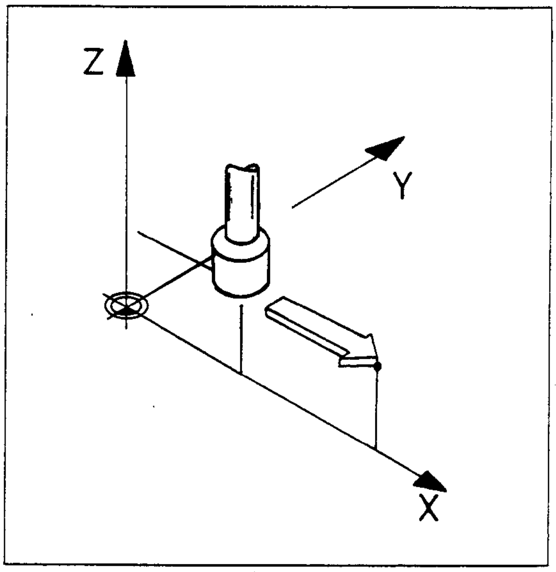
A maximum of three axes can be programmed for straight lines and a maximum of two axes for circles.

Graphics

The examples on the following pages must be supplemented with a uniform BLK FORM if a graphic display is wanted:

```
BLK FORM 0.1   Z   X+0    Y+0    Z-40
BLK FORM 0.2       X+100  Y+100  Z+0
```

Movements are referred to – depending on the number of simultaneously traversed axes – as 1D, 2D or 3D movements (D = dimension).



Single-axis traverse: 1D movements

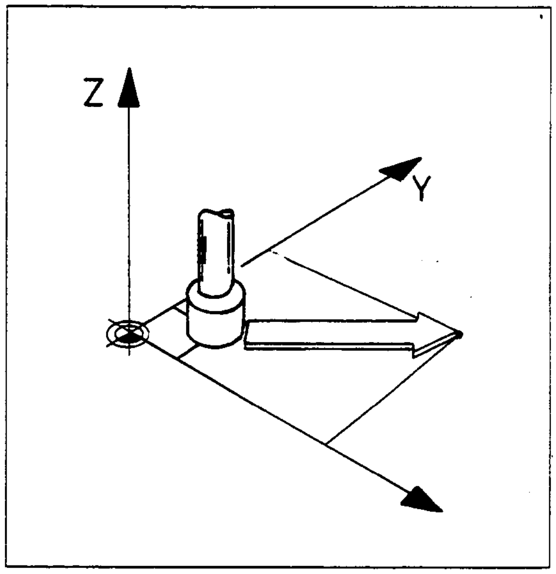
If the electrode is moved relative to the work on a straight line along the direction of a machine axis, this is called single-axis positioning or machining.

Single-axis movements can also be programmed without using the gray path function keys. Only the radius compensation R+/R– is then available (see Radius compensation R+/R–).

2D movements

Movement in a main plane (XY, YZ, ZX) is called 2D movement.

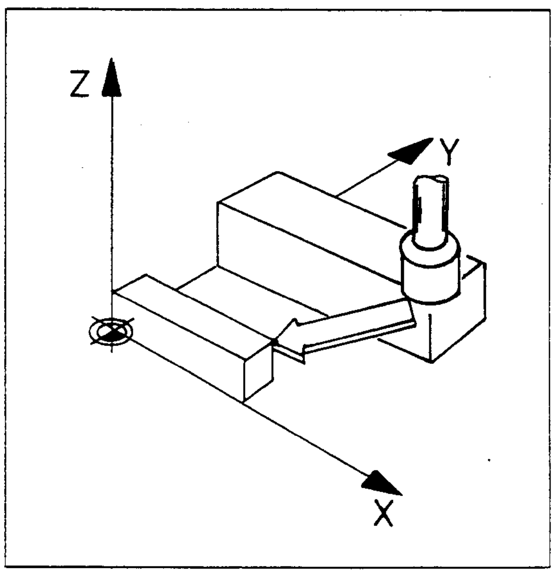
Straight lines and circles can be generated in the main planes with 2D movements.



3D movements

If the electrode is moved relative to the workpiece on a straight line with simultaneous movement of all three machine axes, it is called a 3D straight line.

3D movements are required to generate oblique planes and bodies.



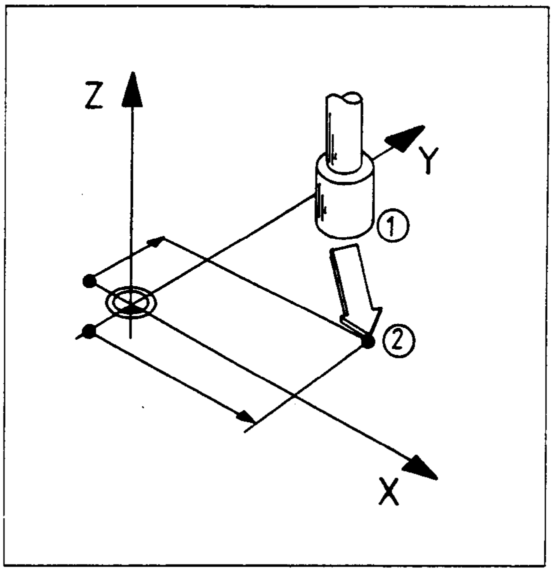
Linear Movement/Cartesian
Positioning in rapid traverse

Positioning

The electrode is at the starting point ① and must travel on a straight line to target point ②.
You always program the target point ② (nominal position) of straight lines.

Position ② can be entered in Cartesian or polar coordinates.

The first position in a program must always be entered as an absolute value. The following positions can also be incremental values.



Example tool definition/ call

TOOL DEF 1 L+10	ENT R4,5 ENT	Electrode 1 has length 10 mm and radius 4.5 mm.
TOOL CALL 1	ENT Z ENT	Electrode 1 is called in the spindle axis Z.
U+1	END □	Undersize is 1 mm.

Positioning block: complete input (main block)

L X+50 Y+30 Z+0 ENT		Z is traversed with length compensation. Only press "ENT" after all simultaneously traversing axes are entered!
	R0 ENT	"R0" is only programmed via "ENT"!
	F NO ENT M36 ENT	Rapid traverse movement "FMAX", erosion on.

L X+50 Y+30 Z+0 R0 F M36

Re-entry at tool calls is especially easy if you enter a main block (= complete positioning block) after a tool call.

Abbreviated input

L X+50 Y+30	END □	Positioning in the XY plane without radius compensation. The electrode is driven to the programmed position (if R0 was programmed in preceding blocks).

After entering the desired values, program blocks may be shortened with the "END □" key if remaining data is unchanged.

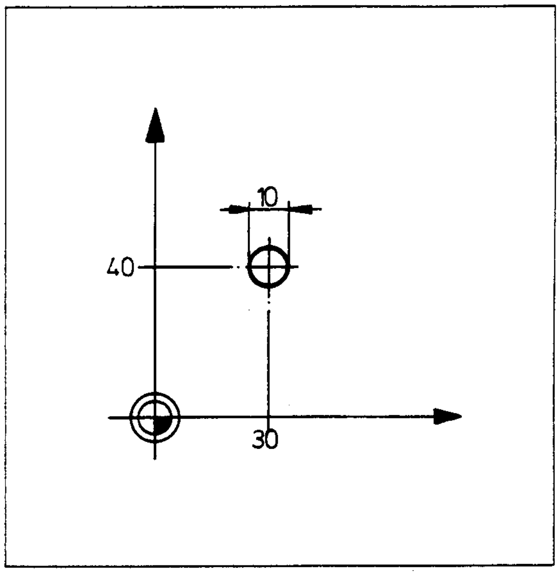
Linear Movement/Cartesian Sinking

Absolute Cartesian coordinates

[L] [X] 30 [Y] 40 [Z] 2 [END]

L X+30 Y+40 Z+2

Multidimensional contour elements can only be entered after initiating with a gray path function key!



Incremental Cartesian dimensions

[L] [I] [X] 20 [END] Only incremental entry.

L IX+20

Mixed entries

[L] [I] [X] 20 [Y] 30 [END] The position for X is entered in incremental dimensions, for Y in absolute dimensions.

L IX+20 Y+30

Example

The cavity described in the following was programmed without using cycles.

Diameter of the cavity: D = 10 mm
Undersize of the electrode according to the erosion parameter table: UM = UNS = 1 mm
Electrode radius:

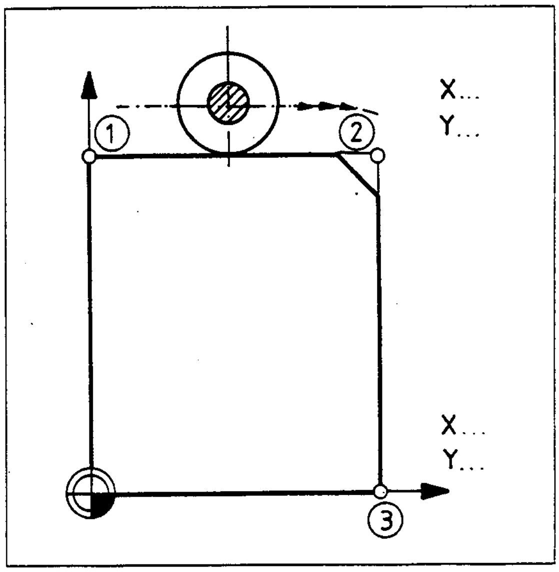
$$Re = \frac{D - UM}{2} = \frac{10\text{ mm} - 1\text{ mm}}{2} = 4.5\text{ mm}$$

Program

0	BEGIN PGM 28 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z–20	Blank form definition (only if graphic workpiece
2	BLK FORM 0.2 X+100 Y+100 Z+0	simulation desired)
3	CYCL DEF 1.0 GENERATOR	Call the desired erosion parameter table for
4	CYCL DEF 1.1 P-TAB 1	generator setting
5	CYCL DEF 1.2 MAX = 3 MIN = 1	Power stage between 3 and 1
6	TOOL DEF 1 L+0 R4.5	Tool definition
7	TOOL CALL 1 Z U+1	Tool call
8	L Z+200 R0 FMAX M6	Retract in Z, tool change
9	L X+20 Y+30 R0 FMAX	Positioning to first hole in X/Y, rapid traverse
10	L Z+2 FMAX	Pre-positioning in Z
11	L Z–10 M36	Sink, erosion on
12	L Z+2 F1000 M37	Retract in Z, erosion off
13	END PGM 28 MM	End of program

Chamfer

A chamfer can be programmed for contour corners formed by the intersection of two straight lines. The angle between the two straight lines can be arbitrary.



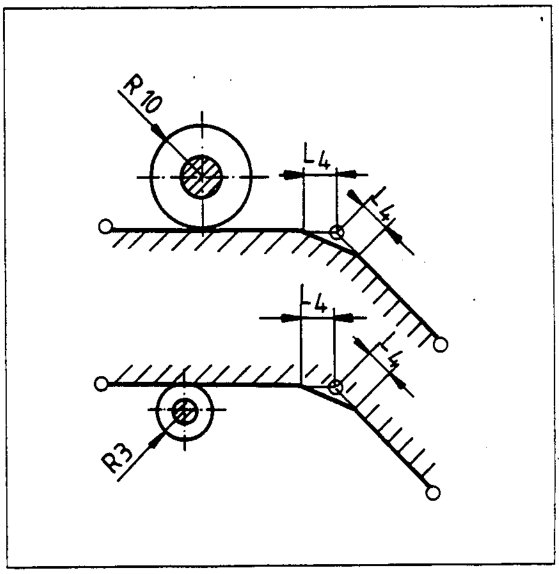
Prerequisites

A chamfer is completely defined by the points ① ② ③ and the chamfer block. A positioning block containing both coordinates of the machining plane should be programmed before and after a chamfer block. The compensation RL/RR/R0 must be identical before and after the chamfer block. A contour cannot be started with a chamfer.

A chamfer can only be executed in the machining plane. The machining plane in the positioning block before and after the chamfer block must therefore be the same.

The chamfer length must not be too long or too short at inside corners: the chamfer must "fit between the contour elements" and also be machineable with the chosen tool.

The previously programmed feed rate remains effective for the chamfer.



Programming

Program a chamfer as a separate block.
Only enter the chamfer length – **no coordinates.**
The "corner point" itself is not traversed!

Entering the chamfer L 4 ENT

L = chamfer length

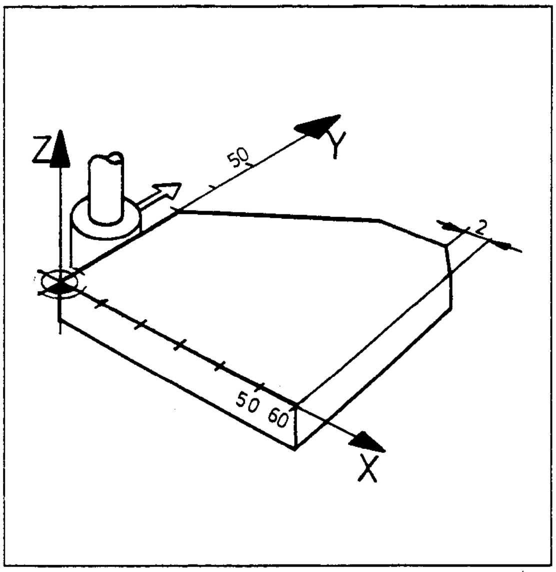
Program block L 4

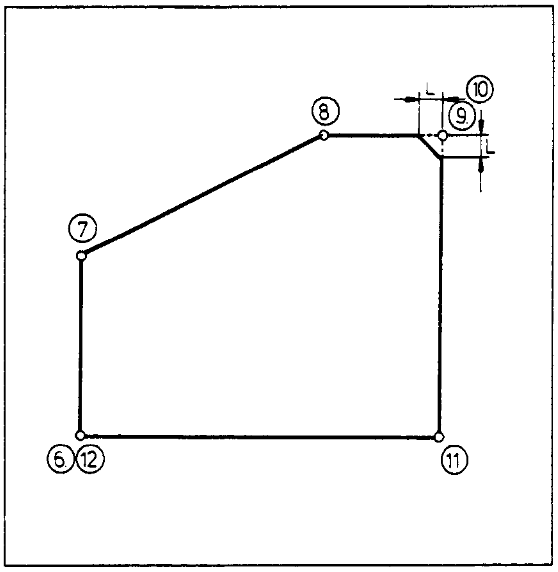
Example

TOOL DEF 1 L+0 R10	
TOOL CALL 1 Z U+0.1	
L X+0 Y+50 RL F300	Position ① (see figure above)
L X+50 Y+50	Position ②
L 4	Chamfer
L X+50 Y+0	Position ③

Linear Movement/Cartesian Example

Example: eroding straight lines





The block numbers are shown in the figure to aid you in following the sequence.

Program

1 CYCL DEF 1.0 GENERATOR	
2 CYCL DEF 1.1 P-TAB 1	
3 CYCL DEF 1.2 MAX = 10 MIN = 10	
4 TOOL DEF 1 L+0 R5	Tool definition
5 TOOL CALL 1 Z U+0.1	Tool call
6 L Z+200 R0 FMAX M6	Tool change
7 L X–10 Y–20 **R0** FMAX M36	Pre-position (tool is up)
8 L Z–20 R F80	Plunge at downfeed rate
9 L X+0 Y+0 **RL** F200	Approach the contour, call radius compensation
10 L X+0 Y+30 RL F400	Machine the contour, switch on erosion
11 L X+30 Y+50 RL	.
12 L X+60 Y+50 RL	.
13 L 2	Chamfer block
14 L X+60 Y+0 RL	
15 L X+0 Y+0 **RL**	Last block with radius compensation
16 L X–20 Y–10 **R0** M37	Cancel radius compensation, switch off erosion
17 L Z+200 R FMAX M2	Back-off Z, return to block 1

Linear axes
U, V, W

Linear interpolation can be performed simultaneously with a maximum of 3 axes – even when using additional axes.

For linear interpolation with an additional linear axis, this axis must be programmed with the corresponding coordinate **in every NC block.** This requirement holds even when the coordinate remains unchanged from one block to the next. If the additional axis is not specified, the control traverses the main axes of the machining plane again.

Example: linear interpolation with X and IV, tool axis Z.

```
11   L   X+0     IV+0    RR F100
12   L   X+100   IV+0
13   L   X+150   IV+70
```

Rotary axes
A, B, C

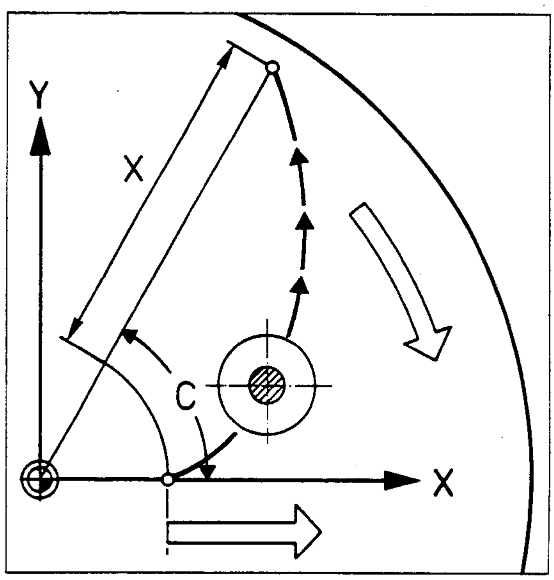
If the additional axis is a rotary axis (A, B or C axis), the control registers the entered value in angular degrees.

During linear interpolation with one linear and one rotary axis, the TNC interprets the programmed feed rate as the path feed rate. That is, the feed rate is based on the relative speed between the workpiece and the tool. Thus, for every point on the path, the control computes a feed rate for the linear axis F_L and a feed rate for the angular axis F_W:

$$F_L = \frac{F \cdot \Delta L}{\sqrt{(\Delta L)^2 + (\Delta W)^2}}$$

$$F_W = \frac{F \cdot \Delta W}{\sqrt{(\Delta L)^2 + (\Delta W)^2}}$$

where:
F = programmed feed rate
F_L = linear component of the feed rate (axis slides)
F_W = angular component of the feed rate (rotary table)
ΔL = linear axis displacement
ΔW = angular axis displacement



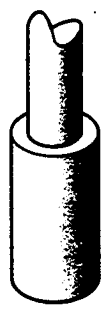
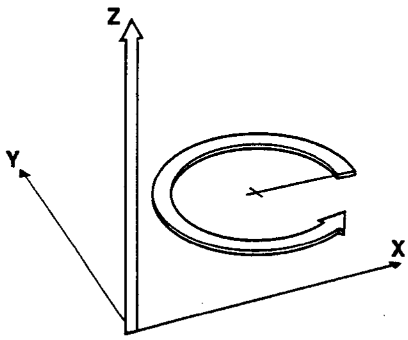
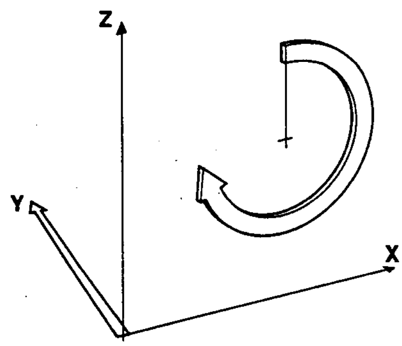
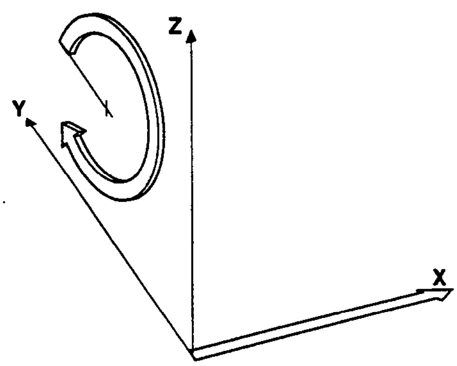
Main planes

Circular arcs can be directly programmed in the main planes XY, YZ, ZX.

TOOL CALL

The circular interpolation plane is selected by defining the spindle axis in the "TOOL CALL" block. This also allocates the tool compensations.
The axis printed bold below (e.g. **X**) is identical in its positive direction with the angle 0° (leading axis).

Interpolation planes

	Spindle axis parallel to	Circular interpolation plane
Standard for milling machines	Z 	XY 
Standard for horizontal borers	Y 	ZX 
	X 	YZ 

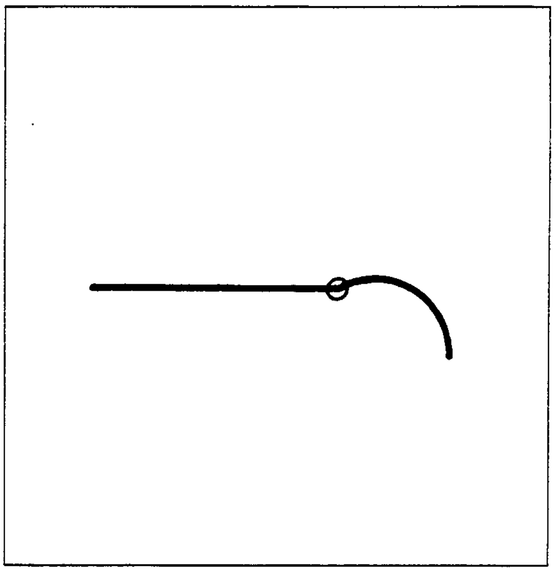
Oblique circles in space

Circular arcs which are not parallel to a main plane can be programmed via Q parameters and executed as a sequence of multiple short straight lines (L blocks).

Circular Movement/Cartesian
Selection guide: Arbitrary transitions

Circular movement
The control moves two axes simultaneously, so the tool describes a circular arc relative to the workpiece.

Arbitrary transitions
The functions C and CR define – together with the preceding block – arbitrary transitions (i.e. tangential and non-tangential transitions) at the beginning and end of the arc.



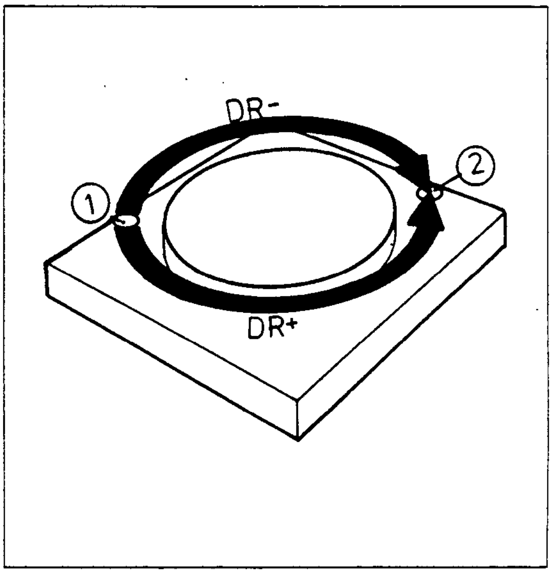
Prerequisite
The starting point ① of the circular movement must be approached in the immediately preceding block.

Circle endpoint
The circle endpoint ② is programmed in a C or CR block.

Rotating direction DR+/DR–
Both definitions also contain the direction of rotation.

Positive rotating direction (in mathematical terms) is **counterclockwise.**

Negative rotating direction is **clockwise.**



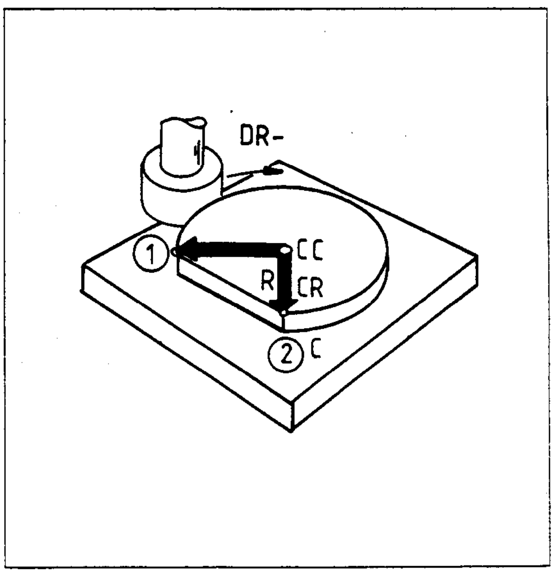
Radius
The radius is indirectly given for "C" as the distance from the position programmed in the immediately preceding C block (start of arc) to the circle center CC.

Full circles
Full circles can only be programmed in one block, with "C".

CR
With CR the radius can be entered directly (CC not required).

Selecting

Given	Select
Starting point of arc ①	e.g. Approach starting point
Circle center	
End point of arc ②	
Starting point of arc ①	e.g. Approach starting point
Radius + end point of arc ②	



Circular Movement/Cartesian
Selection guide: Tangential transitions

Tangential transitions

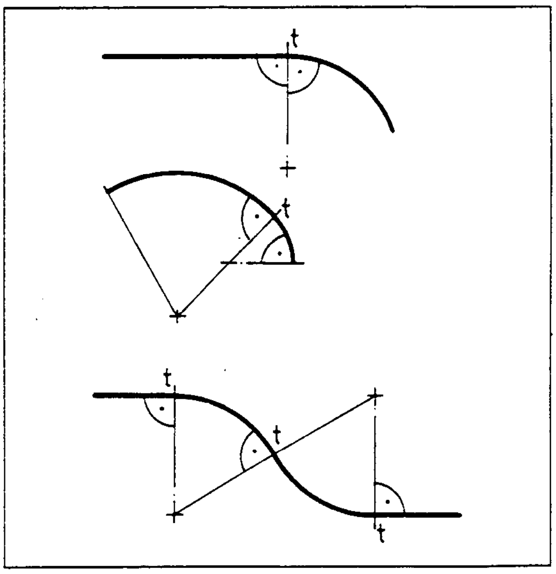
The "RND" and "CT" functions automatically produce a tangential (soft) entry into the arc. Departure from the arc is also tangential with "RND", and arbitrary with "CT". The direction of movement when entering the circle thus also determines the shape of the arc.

Direction of rotation

The direction of rotation need therefore not be given.

Center

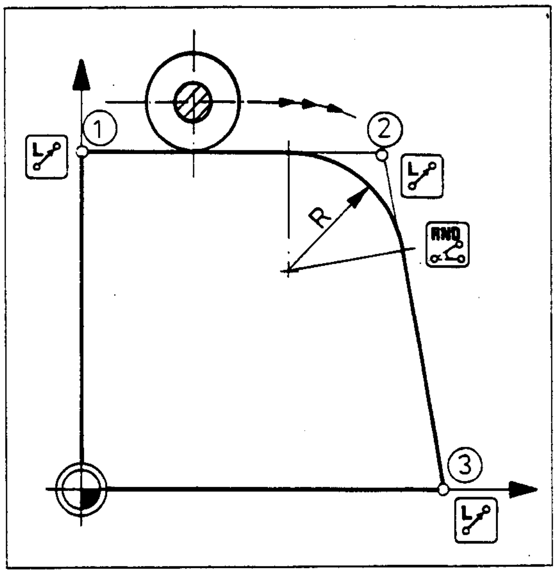
The circle center is not required for either function.



RND

The "RND" rounding is inserted between two contour elements which can be either straight lines or arcs.

Program the **corner point** ② that is not approached and directly thereafter a separate rounding block "RND" with the rounding radius R. Entry and departure from the rounding is necessarily tangential and is automatically computed by the control.

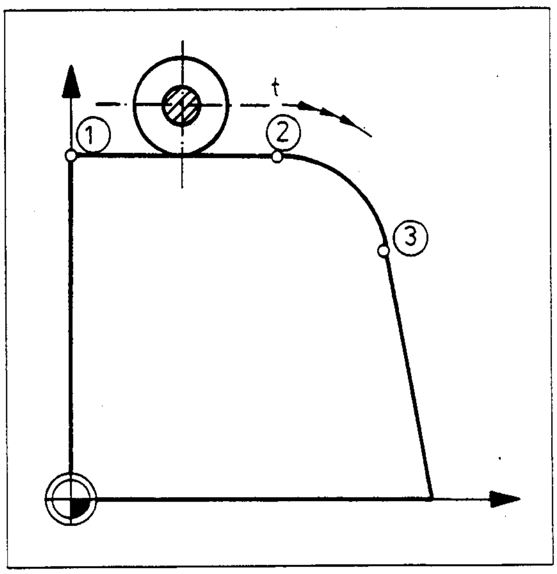


CT

With "CT" **only the arc endpoint** ③ is to be programmed.

Selecting

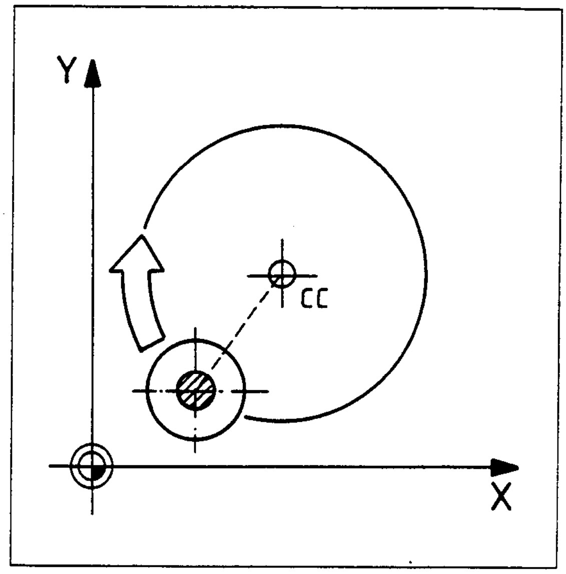
Given	Select
Point ①	e. g. approach with L
Corner ②	e. g. approach with L
Rounding radius	RND
Point ③	e. g. approach with L
Tangent-forming point ①	e. g. approach with L
Tangential entry ②	e. g. approach with L
End point of circular arc ③	CT



Circular Movement/Cartesian CC + C

CC has two functions:

1. Specifying the circle center for circular arcs (to be programmed with "C").
2. Defining the **pole** for polar coordinates.



Circle center CC

The circle center CC must be programmed before circular interpolation with "C". The CC coordinates remain valid until changed by new CC coordinates.

There are three methods for programming CC:

- The circle center CC is directly defined by Cartesian coordinates.
- The coordinates last programmed in a CC block define the circle center.
- The current position is taken as CC with "NO ENT" or "END □" (without numerical input).
  This is also possible for positions programmed in polar coordinates.

The dialog for the circle center is initiated with the "CC" key.

CC absolute: the circle center is based on the work datum.

CC incremental: the circle center is based on the tool position last programmed.

"CC" produces no movement!

Approaching the starting point

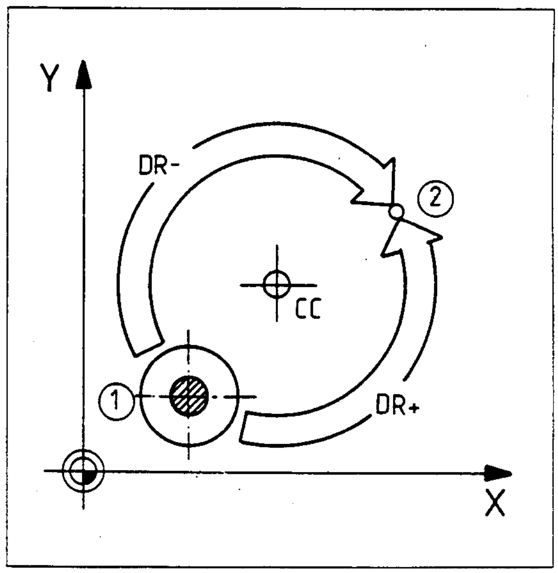
Approach the starting point for the circular arc before the C block.

Radius

The distance from the starting point to the circle center determines the radius.

Circle C

The tool is to travel from position ① to target point ② in a circular path. Only program ② in the C block. Position ② can be entered in Cartesian or polar coordinates.



Direction of rotation

The direction of rotation DR must be defined for circular movement:
rotation in positive direction **DR+** (counterclockwise)
rotation in negative direction **DR–** (clockwise).

Any tool radius compensation must begin before a circular arc.

Tool axis coordinate

Besides the arc end-point coordinates in the working plane, a third coordinate can be entered in the tool axis (e.g. C X+20 Y+20 Z–50).

The tool axis is linearly interpolated with the circular interpolation in the working plane. This feature enables the tool to move in a helical path, for example to approach a contour tangentially in three dimensions.

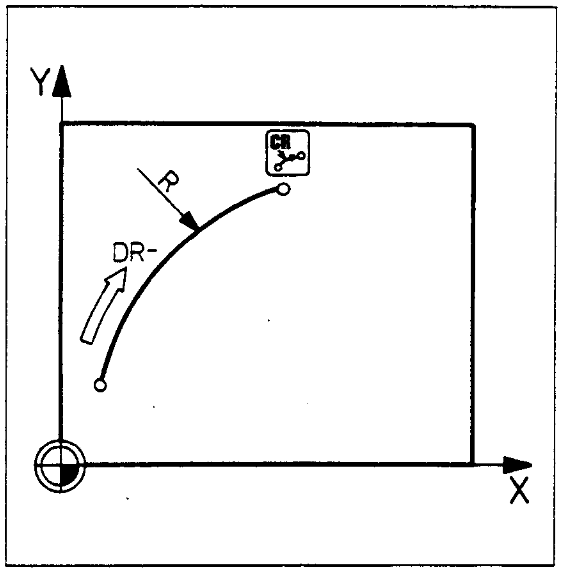
R, F and M are entered as for straight line movements. Input is not necessary unless the values are different from previous input.

Circular Movement/Cartesian CR

Circle CR

If the contour radius is given in the drawing, but no circle center, the circle can be defined via the "CR" key with the
- endpoint of the circular arc
- radius and
- direction of rotation.

R, F and M are entered as for straight lines and are only required when changing earlier specifications.



Starting point

The starting point of the arc must be approached in the preceding block.

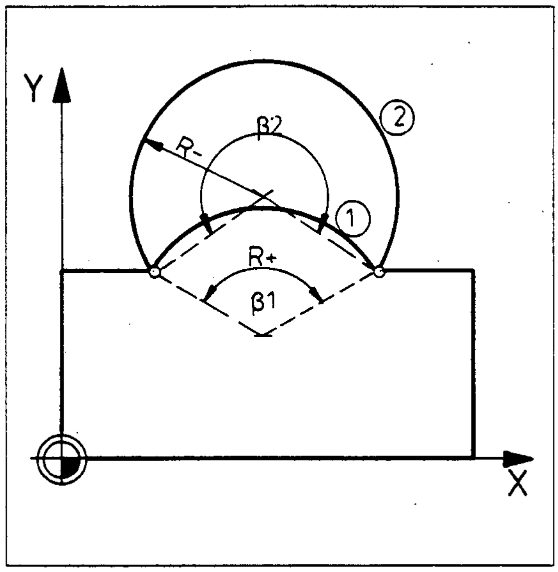
Endpoint

In the CR block the endpoint can only be programmed with Cartesian coordinates.

The distance between starting and end point of the arc must not exceed 2 x R. With CR, full circles can be programmed in 2 blocks.

Central angle

There are two geometric solutions for connecting two points with a defined radius (see figure), depending on the size of the central angle β:
The smaller arc 1 has a central angle $\beta_1 < 180°$, the larger arc 2 has a central angle $\beta_2 > 180°$.



Contour radius

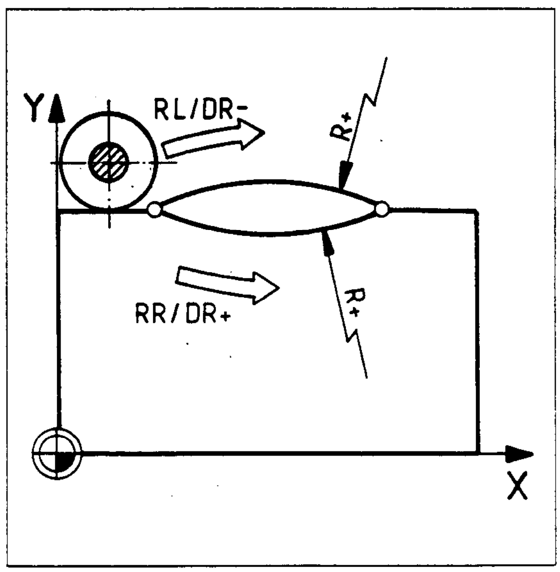
Enter a **positive** radius to program the smaller arc ($\beta < 180°$).
(The + sign is automatically generated.)

To program the larger arc ($\beta > 180°$), enter the radius as a **negative** value.

The maximum definable radius = 30 m.
Arcs up to 99 m can be produced with parametric programming.

Rotating direction

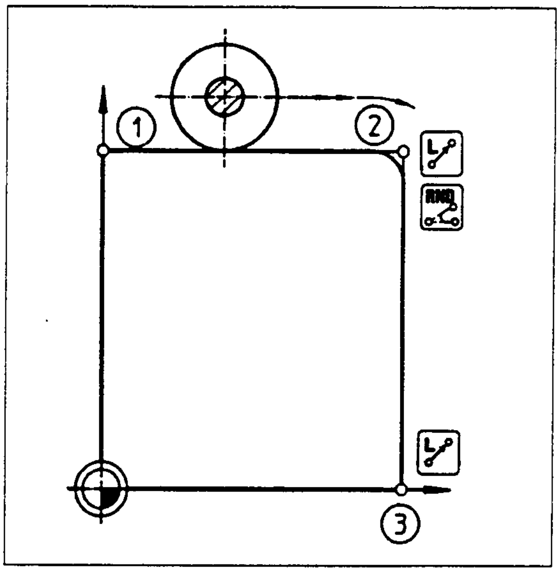
Depending on the allocation of radius compensation RL/RR, the rotating direction determines whether the circle curves inward (= concave) or outward (= convex).
In the adjacent figure, DR– produces a convex contour element, DR+ a concave contour element.



Circular Movement/Cartesian
Corner rounding RND

"RND" has two functions:

- rounding of corners, if RND is "in the contour",
- soft approach and departure from the contour, if RND is at the start or end of the contour.



Circular arc

Contour corners can be rounded with arcs. The circle connects tangentially with the preceding and succeeding contour.

A rounding arc can be inserted at any corner formed by the intersection of the following contour elements:

- straight line – straight line,
- straight line – circle, or circle – straight line,
- circle – circle.

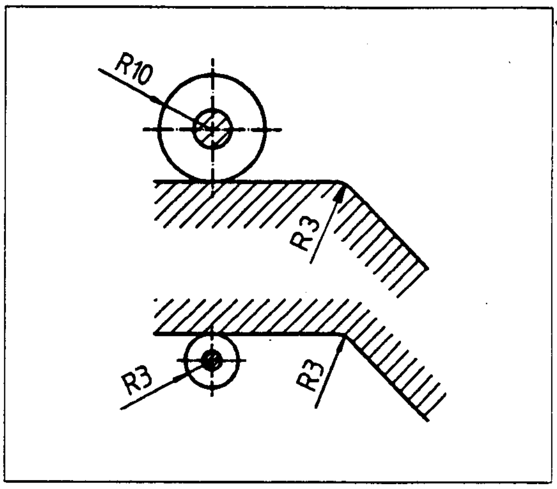
Prerequisites

Rounding is completely defined by the RND block and the points ① ② ③. A positioning block containing both coordinates of the machining plane should be programmed before and after the RND block. The RL/RR/R0 compensation must be identical before and after the RND block.

Note

The rounding arc can only be executed in the machining plane. The machining plane must be the same in the positioning block before and after the rounding block.

The rounding radius cannot be too large or too small for inside corners – it must "fit between the contour elements" and be machinable with the current tool.



The electrode radius can be larger than the rounding radius on outside corners.

The electrode radius must be smaller than or equal to the rounding radius on inside corners.

Programming

The rounding arc is programmed as a separate block following the corner to be rounded. Enter the rounding radius and a reduced feed rate F, if needed. The "corner point" itself is not approached!

Error messages

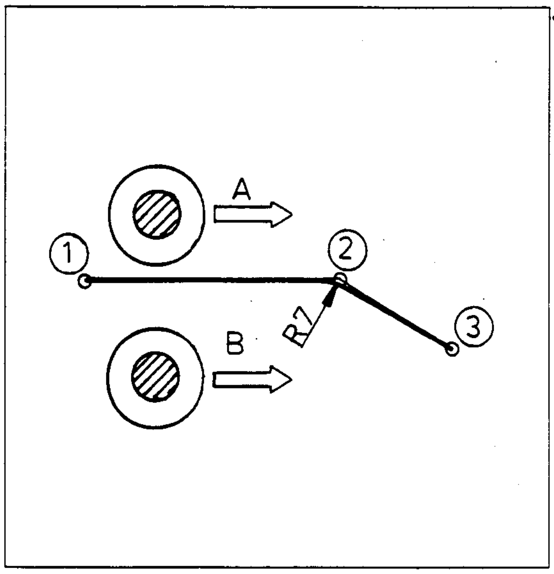
PLANE WRONGLY DEFINED
The machining planes are not identical before and after the RND block.

ROUNDING RADIUS TOO LARGE
The rounding is geometrically impossible.

Circular Movement/Cartesian
Corner rounding RND

Input RND	RND 8	END	Rounding radius
Program block	RND 8		

Examples:	TOOL DEF 1 L+0 R5 TOOL CALL 1 Z U+0.1	
Sequence A	L X+10 Y+60 RL M36	Position ①
	L X+50 Y+60	"Corner point" ②
	RND 7	Rounding
	L X+90 Y+50	Position ③
Sequence B	L X+10 Y+60 RR M36	Position ①
	L X+50 Y+60	"Corner point" ②
	RND 7	Rounding
	L X+90 Y+50	Position ③



Circular Movement/Cartesian
Tangential arc CT

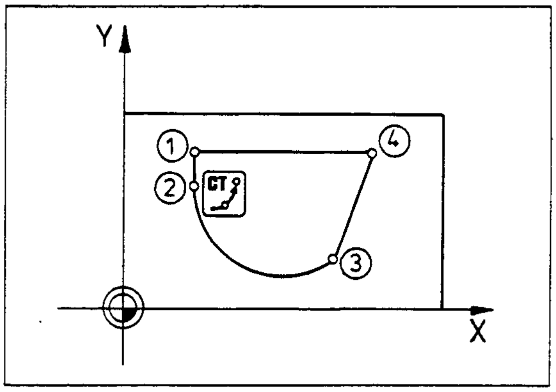
Circular arc CT

A circular arc can be programmed more easily if it connects tangentially to the preceding contour. The circular arc is defined by **merely entering the arc endpoint** with the "CT" key.

Geometry

An arc with tangential connection to the contour is exactly defined by its endpoint.

This arc has a specific radius, a specific direction of rotation and a specific center. This data need not therefore be programmed.



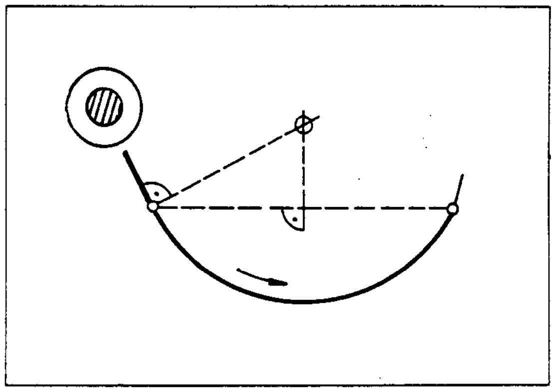
Machining sequence

Prerequisites

The contour element which connects tangentially to the circle is programmed immediately before the tangential arc. Both coordinates of the same machining plane must be programmed in the block for the tangential arc and in the preceding block.

Tangent

The tangent is specified by **both** positions ① and ② directly preceding the CT block. Therefore, the first CT block can appear no earlier than the third block in a program.

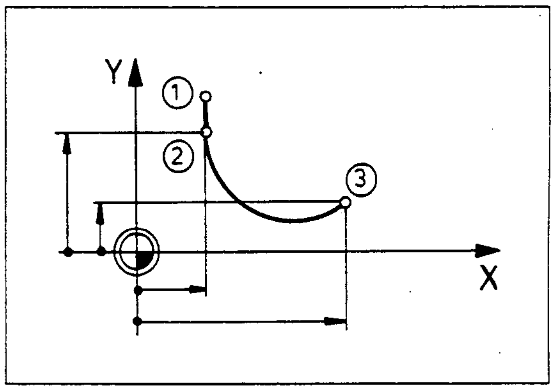

Geometry

Circular arc CT

The electrode is to travel a circle connecting tangentially to ① and ② to target point ③. Only ③ is programmed in the CT block.

Coordinates

The endpoint of the circular path can be programmed in either Cartesian or polar coordinates.



Cartesian coordinates

Error messages

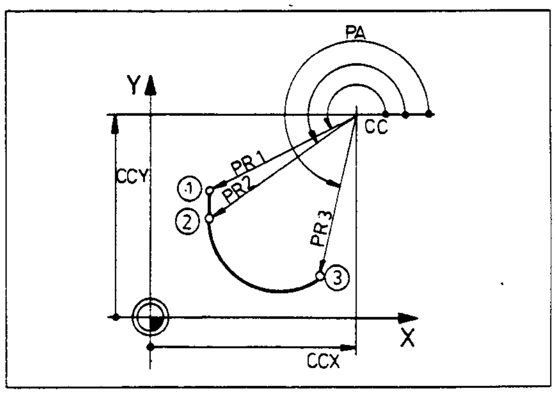
WRONG CIRCLE DATA
The required minimum 2 positions before the CT block were not programmed.

ANGLE REFERENCE MISSING
Both coordinates of the machining plane are not given in the CT block and the preceding block.

Tool axis coordinate

Besides the arc end-point coordinates in the working plane, a third coordinate can be entered in the tool axis (e.g. C X+20 Y+20 Z–50 or CT X+90 Z+27 IC+20).

The tool axis is linearly interpolated with the circular interpolation in the working plane. This feature enables the tool to move in a helical path, for example to approach a contour tangentially in three dimensions.



Polar coordinates

Circular Movement/Cartesian
Tangential arc CT

Input CT CT X 90 Y 40 ENT — Arc endpoint

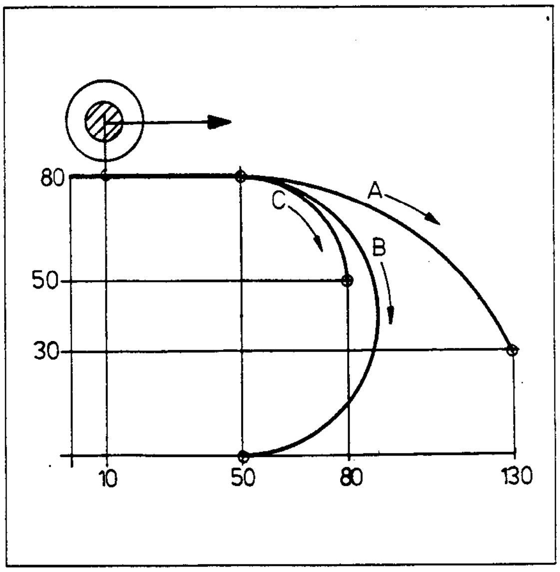
Program block CT X+90 Y+40

Enter R, F and M as for straight lines.
Input is only necessary to change earlier definitions.

Examples: different endpoints

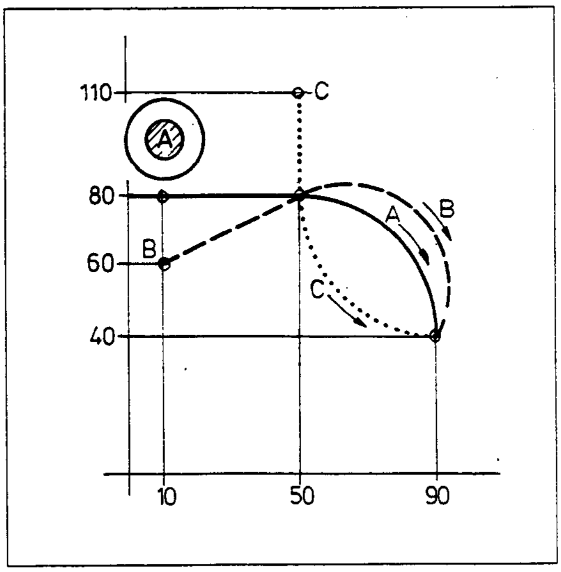
TOOL DEF 1 L+0 R10
TOOL CALL 1 Z U+0,5

Arc A	L X+10 Y+80 RL M36	1st tangent point
	L X+50 Y+80	Start of arc
	CT X+130 Y+30	End of arc
Arc B semicircle	L X+10 Y+80 RL M36	1st tangent point
	L X+50 Y+80	Start of arc
	CT X+50 Y+0	End of arc. A semicircle with R = 40 is formed.
Arc C quarter circle	L X+10 Y+80 RL M36	1st tangent point
	L X+50 Y+80	Start of arc
	CT X+80 Y+50	End of arc. A quarter circle with R = 30 is formed.



Different tangents

Arc A	L X+10 Y+80 RL M36
	L X+50 Y+80
	CT X+90 Y+40
Arc B	L X+10 Y+60 RL M36
	L X+50 Y+80
	CT X+90 Y+40
Arc C	L X+50 Y+110 RL M36
	L X+50 Y+80
	CT X+90 Y+40

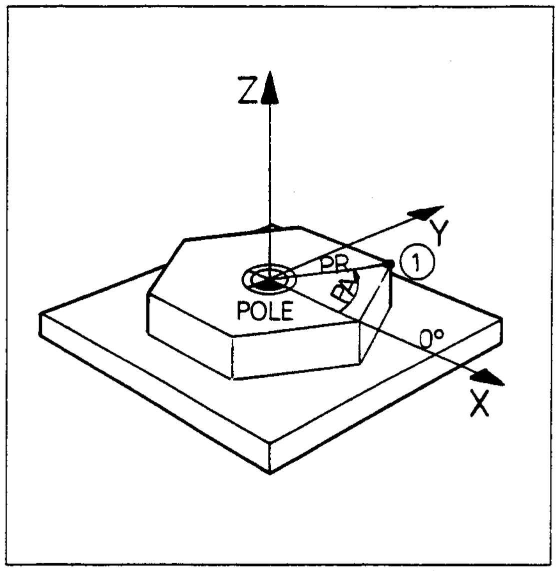


Polar Coordinates Fundamentals

The control allows you to enter nominal positions in either Cartesian or polar coordinates.

In polar coordinates, the points in a plane are specified by the **polar radius PR** (distance to the pole), and the **polar angle PA** (angular direction).

The pole position is entered with the "CC" key in Cartesian coordinates based on the workpiece datum.



Marking

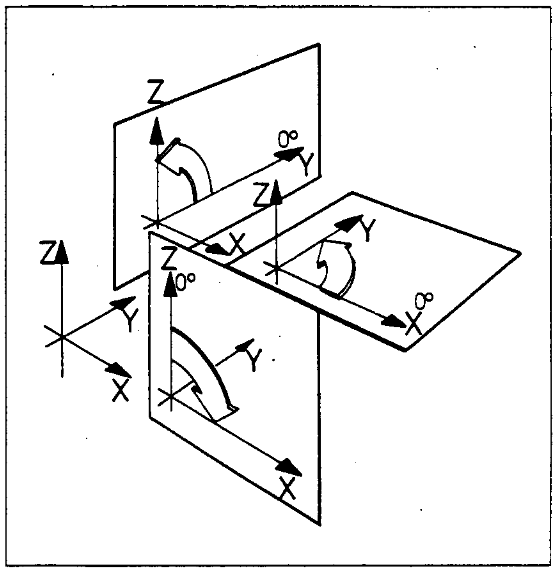
Blocks in polar coordinates are marked by a P (LP, CP etc.).

Angle reference axis

The angle reference axis (0° axis) is the
+X axis in the XY plane,
+Y axis in the YZ plane,
+Z axis in the ZX plane.

The machining plane (e. g. XY plane) is determined by a tool call.

The sign of the angle PA can be seen in the adjacent figure.



Absolute polar coordinates

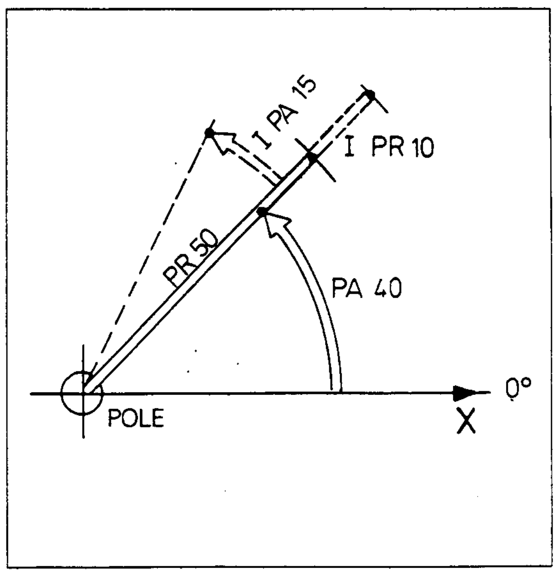
Absolute dimensions are based on the current pole.
Example: LP PR+50 PA+40

Incremental polar coordinates

A polar coordinate radius entered incrementally changes the last radius.
Example: LP IPR+10

An incremental polar coordinate angle IPA refers to the last direction angle.
Example: LP IPA+15

Absolute and incremental coordinates may be mixed within one block.
Example: LP PR+50 IPA+15



Polar Coordinates
Pole

Pole

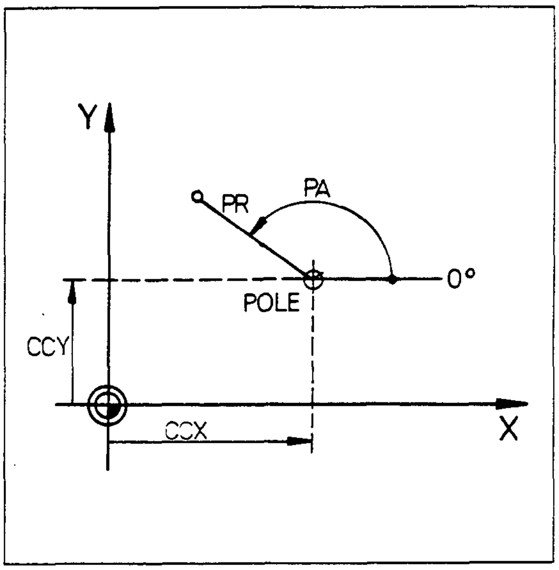
The pole must be specified with "CC" before entering polar coordinates. The pole can be set anywhere in the program prior to using polar coordinates.

The pole is programmed in absolute or incremental Cartesian coordinates.

CC absolute: the pole is referenced to the workpiece datum.

CC incremental: the pole is referenced to the last programmed nominal tool position.

A CC block is programmed with the coordinates of the machining plane.



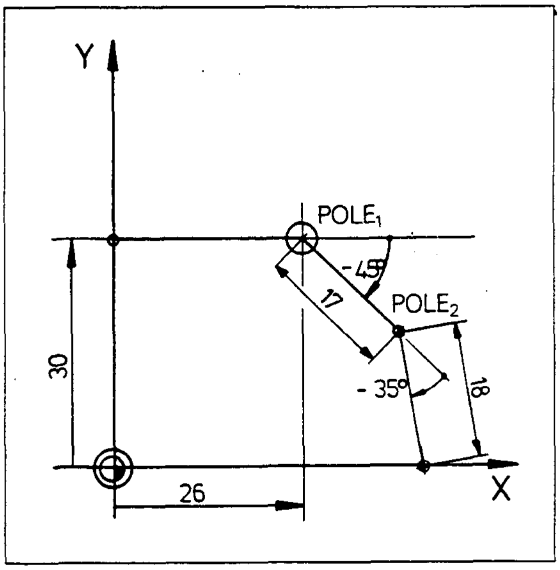
Example

CC X+60 Y+30

Transferring the pole

The last **programmed position** is **transferred** as the pole. Program an empty CC block.

Directly transferring the pole in this manner is especially well suited for polygon shapes with polar dimensions (see illustration below).

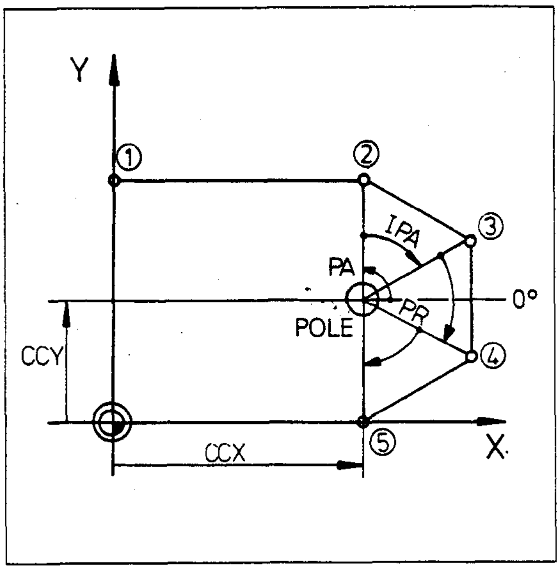


Example

L X+26 Y+30	
CC	POLE 1
LP PR+17 PA–45	
CC	POLE 2
LP PR+18 IPA–35	

Modal effect

A pole definition remains valid in a program until it is overwritten with another definition. The same pole therefore need not be programmed repeatedly.



Polar Coordinates
Straight line LP

After opening with the "L" key, you must press the "P" key to enter positions in polar coordinates.

For dimensions which are referenced to a rotational axis in some way, such as bolt hole circles or cams, programming is usually easier in polar coordinates than in Cartesian coordinates because calculations are avoided.

Third, Cartesian coordinate

In an LP block, a third, Cartesian coordinate can be entered in the tool axis besides the polar radius PR and the polar angle PA in the working plane (e.g. **LP PR+40 PA+200 IZ–10 RR**).

Range for polar angle PA

Input range for linear interpolation: absolute or incremental –360° to +360°.

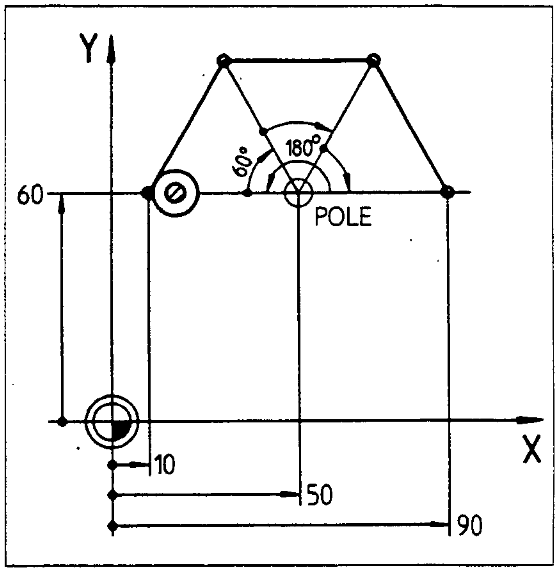
PA positive: counterclockwise angle.
PA negative: clockwise angle.

Example

Eroding an inside contour:

Program

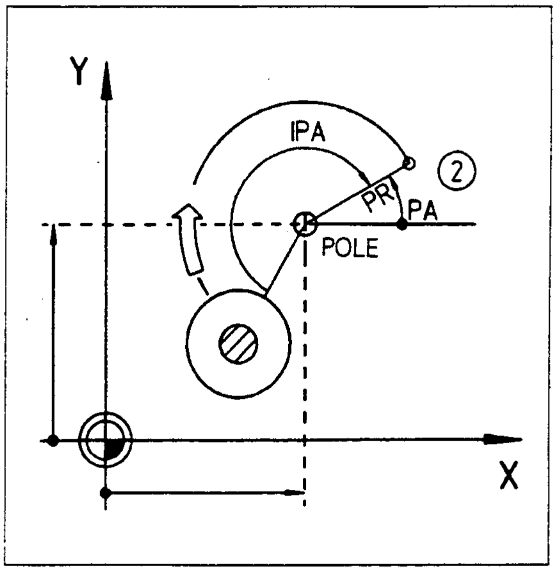
TOOL DEF 2 L+0 R2 TOOL CALL 2 Z U+0,5	
CC X+50 Y+60	Set POLE
L X+15 Y+50 R0 F1000	Approach starting point externally (Cartesian coordinates)
L Z–5 M36	Plunge
LP PR+40 PA+180 RR	Approach 1st contour point with compensation (polar coordi nates)
LP IPA–60	2nd contour point
LP IPA–60	
LP IPA–60	Last contour point
L X+85 Y+50 R0 M37	Depart from contour, cancel compensation
L Z+50 R0 FMAX M2	Retract



Polar Coordinates
Circular path CP

Circular arc

If the target point on the arc is programmed in polar coordinates, you only have to enter the polar angle PA to define the endpoint. The radius is defined by the distance from the starting point of the arc to the programmed circle center CC.



When programming a circle in polar coordinates, the angle PA and the rotating direction DR can be entered positively or negatively. The angle PA determines the endpoint of the arc.

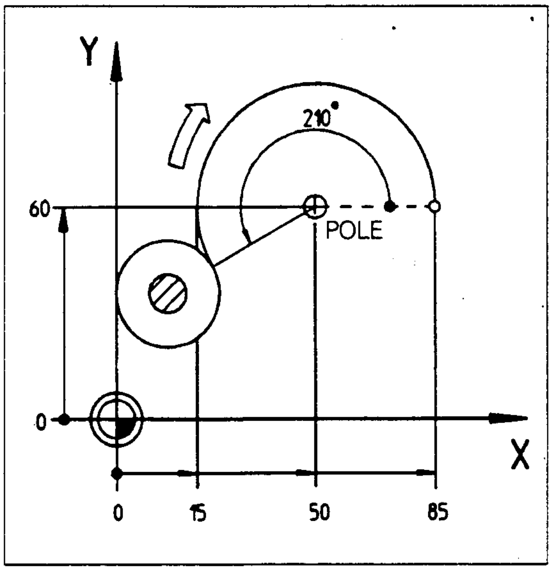
If the angle PA is entered incrementally, the sign of the angle and the sign of the rotating direction should be the same. In the figure to the right, this means that IPA is negative and DR is also negative.

Range for polar angle PA

Input range for circle interpolation:
absolute or incremental −5400° to +5400°.

Example

An arc with radius 35 and circle center X+50 Y+60 is to be eroded.
Rotating direction is clockwise.



Program

TOOL DEF 1 L+0 R5
TOOL CALL 1 Z U+0,5

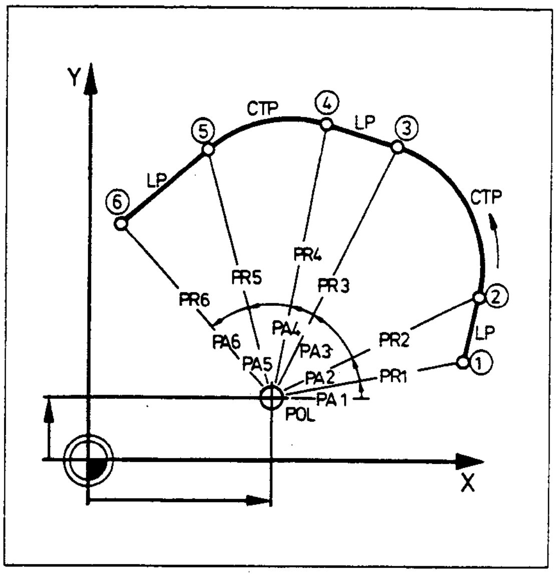
CC X+50 Y+60	Coordinates of circle center
LP PR+35 PA+210 RL F200	Approach circle (circle radius is 35 mm)
CP PA+0 DR- M36	Circular movement clockwise

In the example, a contour radius of 35 mm is obtained from the distance between the POLE and the approach point on the circle.

Polar Coordinates
Tangential arc CTP
Corner rounding RND

Tangential arc

The endpoints of tangential arcs may be entered in polar coordinates to simplify the programming of, for example, cams.



Third, Cartesian coordinate

In the connecting circular block CTP, a third, Cartesian rotary coordinate in the tool axis can be entered besides the polar radius PR and the polar angle PA in the working plane (e.g. **CTP PR+30 PA+900 IC –5**). This feature enables the tool to move in a helical path, or the C axis to move tangentially to an arc to retain the orientation of the tool to the workpiece contour.

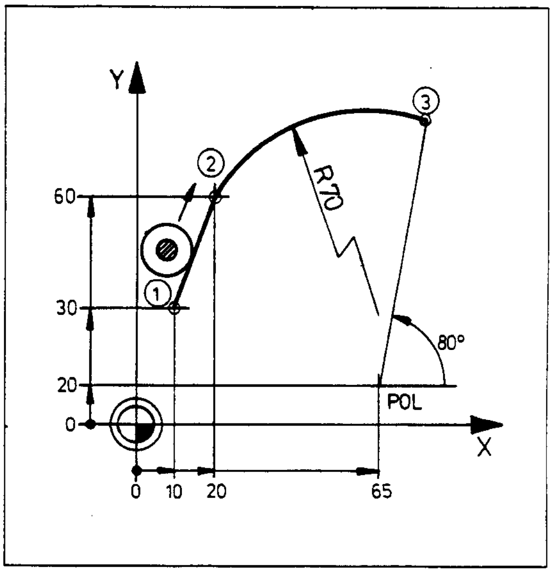
The start of the arc is automatically tangential when programming with CT in the working plane.

If the transition points are not calculated exactly, the arc elements could become "jagged".

Specify the pole CC before programming in polar coordinates.

Example

A straight line through ① and ② is to tangentially meet the arc to ③. The radius and direction angle of ③ with respect to CC are known.



Program

TOOL DEF 1 L+0 R4
TOOL CALL 1 Z U+0.5

CC X+65 Y+20

L X+10 Y+30 RL M36

L X+20 Y+60

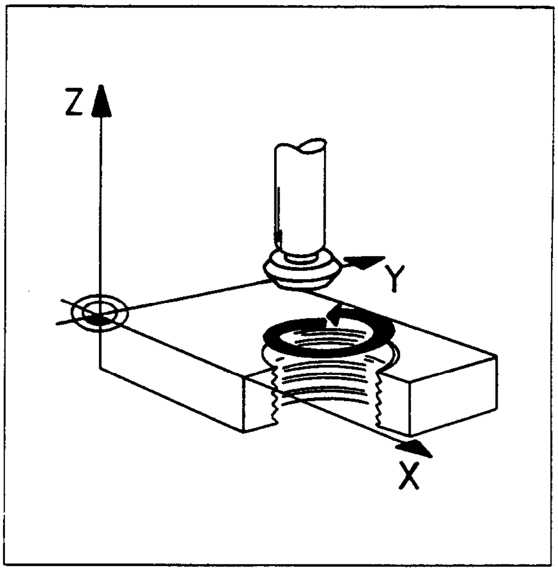
CTP PR+70 PA+80

Polar "corners" can also be rounded with the "corner rounding" function (see Circular Movement/Cartesian, Corner rounding RND).

Polar Coordinates
Helical interpolation (CC + CP) + Z

Helix

If 2 axes are moved simultaneously to describe a circle in a main plane (XY, YZ, ZX), and a uniform linear motion of the tool axis is superimposed, then the tool moves along a helix (helical interpolation).



Applications

Helical interpolation can be used to advantage with form electrodes for producing internal and external threads with large diameters. This can save you substantial tool costs.

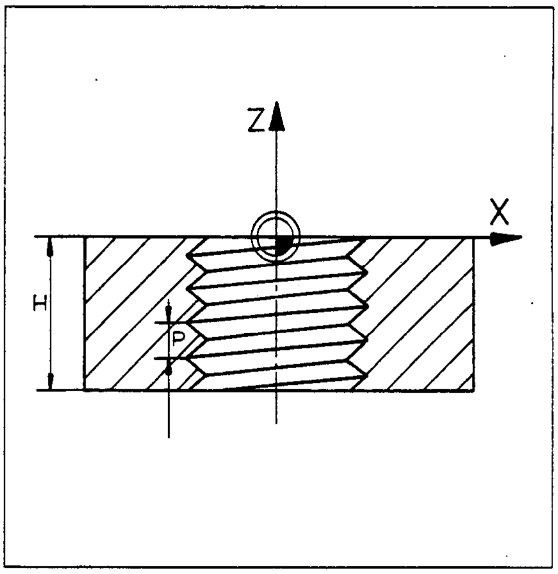
Input data

The helix is programmed in polar coordinates.

First specify the POLE or circle center CC.

Angle range

Enter the total angle of tool rotation for the polar angle IPA in degrees:
IPA = number of rotations x 360°
Maximum angle of rotation: ± 5400° (15 complete rotations).



Height

The total height H (= IZ) is entered for the tool axis (Z) at the query "Coordinates".

Calculate the value from the thread pitch and the required number of tool rotations.
IZ = P · n, IZ = total height/depth to be entered
P = pitch
n = number of threads
The total height/depth can be entered in absolute or incremental dimensions.

Thread

A complete thread can be programmed quite easily with IZ and IPA; the number of threads is then specified with a program section repeat REP.

Radius compensation

The radius compensation depends upon the

- rotating direction (right/left),
- type of thread (internal/external),
- erosion direction (positive/negative axis direction)
  (see table to the right).

Internal thread	Working direction	Rotating direction	Radius compensation
right-hand	Z+	DR+	RL
left-hand	Z+	DR–	RR
right-hand	Z–	DR–	RR
left-hand	Z–	DR+	RL

External thread	Working direction	Rotating direction	Radius compensation
right-hand	Z+	DR+	RR
left-hand	Z+	DR–	RL
right-hand	Z–	DR–	RL
left-hand	Z–	DR+	RR

Polar Coordinates
Helical interpolation (CC + CP) + Z

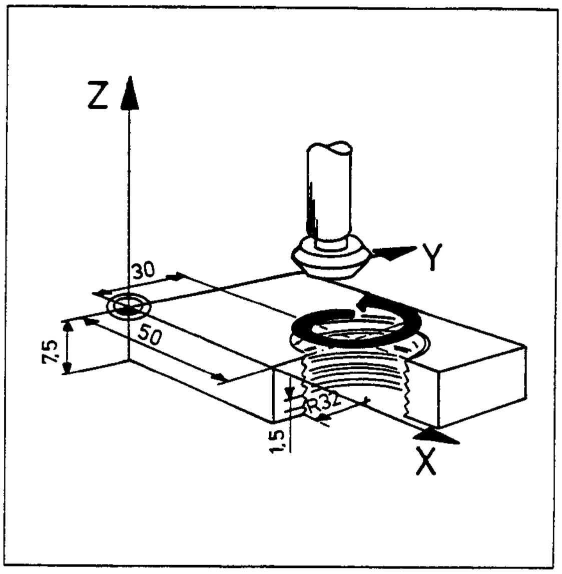
C P + Z

Input example

CP P I 360 I Z 2 — Endpoint

+/ +/ END — Rotating direction

CP IPA+360 IZ+2 DR+



Task

A right-hand internal thread M64 x 1.5 is to be produced in one pass with a special electrode.

Thread

Thread data:

pitch	$P = 1.5$ mm
start	$\alpha_a = 0°$
end	$\alpha_e = 0° = 360°$
Number of threads	$n_0 = 5$
Overrun of threads:	
at start	$n_1 = 1/2$
at end	$n_2 = 1/2$

Calculations

Total height:
$IZ = P \cdot n = 1.5\text{ mm} \cdot [5 + (2 \cdot 1/2)] = 9\text{ mm}$

Incremental polar angle of orbit:
$IPA = 360° \cdot n = 360° \cdot [5 + (2 \cdot 1/2)] = 2160°$

Due to overrun of 1/2 thread, the start of thread is advanced by 180°:
starting angle $\alpha_s = \alpha_a + (-180°) = 0° + (-180°) = -180°$

The overrun of 1/2 thread at the start of thread gives the following initial value for Z:
$Z = -P \cdot n = -1.5\text{ mm} \cdot [5 + 1/2] = -8.25\text{ mm}$

Program

TOOL DEF 1 L+0 R19,8 TOOL CALL 1 Z U+0,4	
L X+50 Y+30 R0 FMAX	Approach the hole center
CC	Take the position as pole
L Z–8,25 R0 FMAX	Downfeed at center to **initial value Z**
LP PR+32 PA–180 RL F100	Approach the wall with radius R and **starting angle** α_s
CP IPA+2160 IZ+9 DR+ RL M36	Helical movement with **incremental angle IPA** and **total height IZ**
L X+50 Y+30 M37	Retract in XY
L Z+100 FMAX	Retract in Z

Note

Helical interpolation cannot be graphically displayed.

Polar Coordinates
Circular interpolation (CC + CP) + C with linear interpolation in the C axis

Definition

Two axes are moved simultaneously such that a circle is described in the XY working plane. This movement is combined with a rotation (linear interpolation) of the C axis.

Application

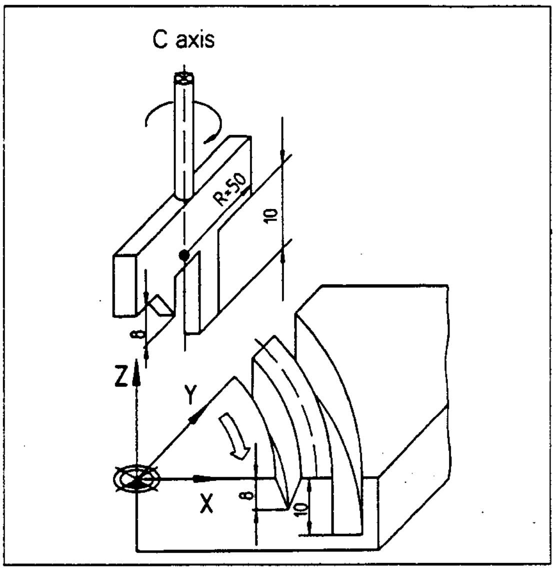
The angular orientation of the electrode is very important for eroding circular paths with non-cylindrical electrodes. The position of the electrode in the circular arc can the followed very exactly with the aid of linear interpolation of the C axis with a circular interpolation in XY.

Input data

The coordinates of the main plain XY and the C axis must be entered in polar coordinates after the pole CC has been entered.

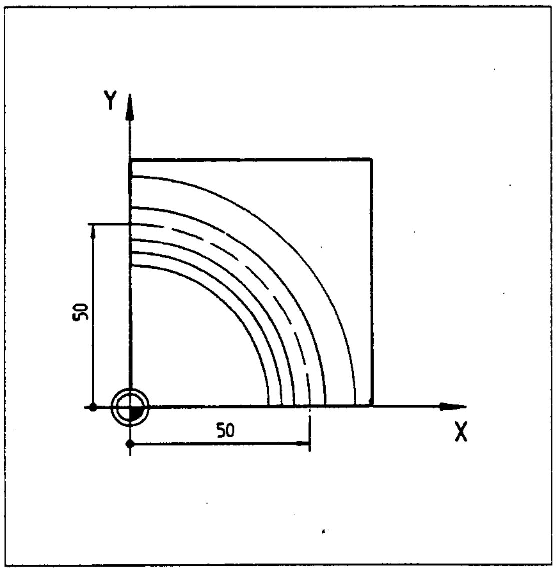
Task

Two adjacent slots which describe a quarter circle are to be eroded with the aid of an especially shaped electrode.



Program

```
TOOL DEF 1 L+0 R+50
TOOL CALL 1 Z U+0.5
L X+0 Y+50 R0 FMAX   Approach starting point
CC X+0 Y+0           Set the pole
L Z–10 R0 M36        Sink
CP IPA–90 IC–90      Erode the quarter circle
DR– R0
L Z+100 FMAX M37     Retract in the Z direction
```



Polar Coordinates
Circular interpolation (CC + CP) + Z + C with linear interpolation in the C axis

Definition

Two axes are moved simultaneously such that a circle is described in the main plain XY. This movement is combined with a linear interpolation in the Z and C axes (this function is not available in the export version TNC 306 E).

Application

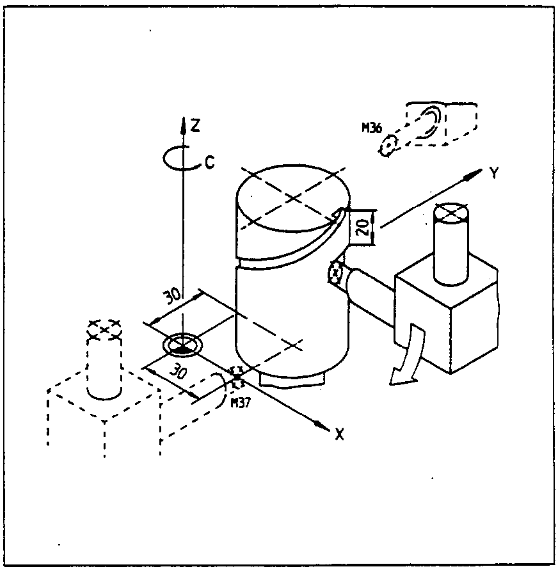
With this interpolation it is possible, for example, to machine lubrication grooves on a helical path using a form electrode.

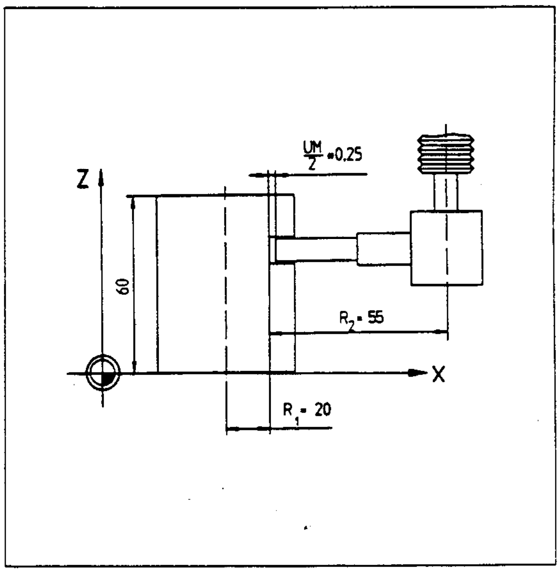
Input data

The coordinates of the main plane XY, of the Z axis and of the C axis must be entered in polar coordinates after the pole CC has been entered.

Task

A helical lubrication groove is to be eroded into a cylindrical surface with the aid of a form electrode (function not available in the export version TNC 306 E).





Program

TOOL DEF 1 L+0 R+0	
TOOL CALL 1 Z U+0.5	
L Z+100 R0 M6	Change electrode
L X+30 Y+105.25	Starting position
L Z+65 C–90	Approach
CC X+30 Y+30	Set the pole
CP IPA–1260 IZ-70 IC-1260 M36	Helical interpolation with the incremental polar angel IPA; the total height IZ, **erosion on**
L Z–15 IC+90 M37	Retract electrode, **erosion off**
L X–50 Y–50	Retract in XY
L Z+100	Retract in Z

Polar Coordinates
Circular interpolation (CC + CP) + Z + C with linear interpolation in the C axis

Lubrication groove

Pitch $P = 20$ mm

Beginning $\alpha_a = 0°$

End $\alpha_e = 0°$ (360°)

Number of revolutions: $n_0 = 3$

Overrun for each revolution:
at start: $n_1 = 1/4$
at end: $n_2 = 1/4$

Calculations

Total height:
$IZ = P \cdot -[n_0 + n_1 + n_2] = 20 \text{ mm} \cdot -[3 + 1/4 + 1/4] = -70 \text{ mm}$

Incremental angle of traverse for polar-circular interpolation and C axis:
$IPA = 360° \cdot (-n) = 360° \cdot (-3.5) = -1260°$

The beginning of the lubrication groove is advanced by 90° through the overrun of 1/4 revolution:

Starting position $Z_S = P \cdot [n_0 + n_1] = 20 \text{ mm} \cdot [3 + 1/4] = 65 \text{ mm}$

$$X_S = X_{CC} = 30 \text{ mm}$$

$$Y_S = Y_{CC} + R_1 + R_2 + \frac{UM}{2} = 105.25$$

$$C_S = \alpha_a + 90° = 0° + 90° = 90°$$

Contour Approach and Departure
Starting and end position

Selecting the 1st contour point Before beginning contour programming, specify the first contour point at which machining **with radius compensation** is to begin.

Starting point In the vicinity of the first contour point, define an **uncompensated starting point** that can be approached in rapid traverse, and be sure to consider the tool in use. The starting point must fulfill the following criteria:

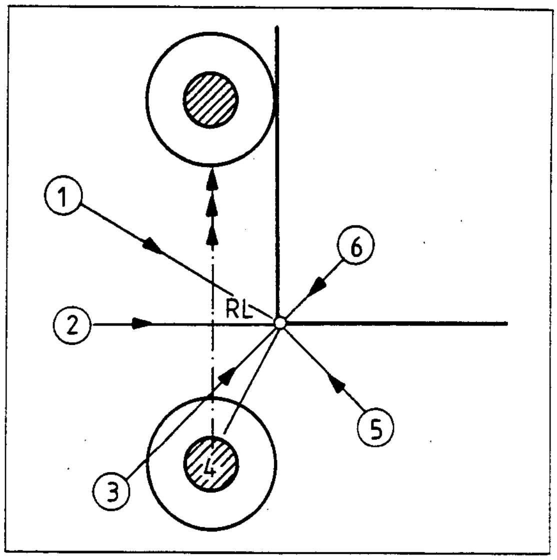
- approachable without collision
- near the first contour point
- outside the material
- the contour will not be damaged when approaching the first contour point.

Direct approach When working on a circle (RND) without the TNC approach/departure function, also check that the electrode does not distort the contour due to a direction change.

Starting points

① Not recommended	Surface blemish due to change of Y-axis direction
② Not recommended	
③ **Suitable**	**Also for end point**
④ **Optimal**	Lies on the extension of the compensated path
⑤ Not recommended	Contour damage
⑥ Not permitted!	

Radius compensation must remain switched off for the starting position (R0).



End points The same prerequisites apply for selecting the **uncompensated end point** as for the starting point.

The ideal end point ④ lies on the extension of the last contour element RL.

①, ② Not recommended	A burr is left due to change of the X-axis direction
③ **Suitable**	**Also for the starting point**
④ **Optimal**	Lies on the extension of the compensated path
⑤ Not recommended	Contour damage
⑥ Not permitted!	

Radius compensation must be switched off after departure from the contour (R0).

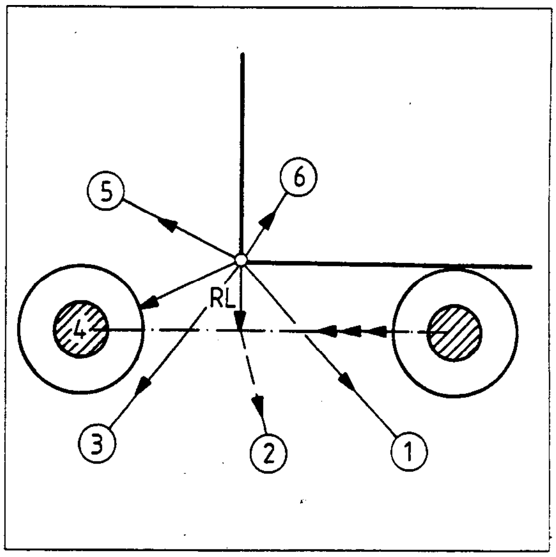


Illustration
——— programmed path
— · — · — traversed electrode center path

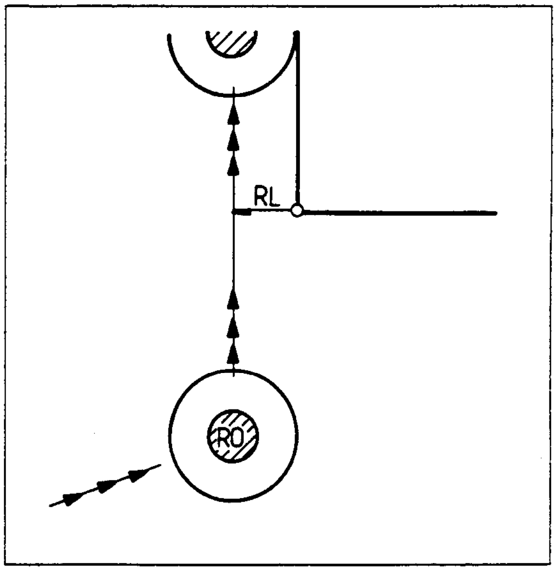
Common starting and end point For a **common starting and end point**, select point ③ on the bisecting line of the angle between the first and last contour element.

Contour Approach and Departure
Starting and end position

Approach

The starting position Ⓢ must be programmed without radius compensation, i. e. with R0.

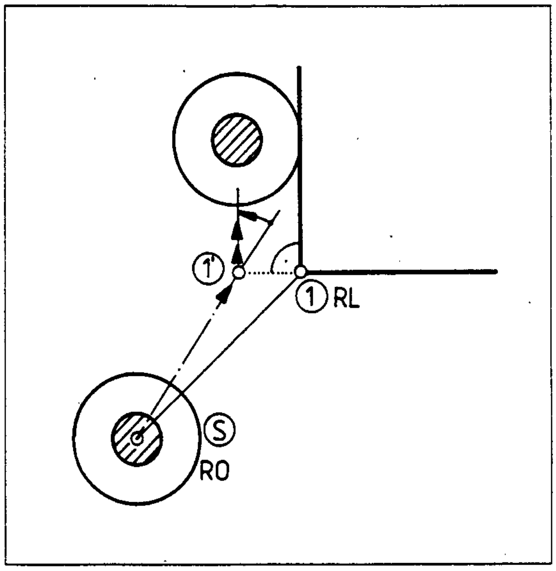
The control guides the electrode in a straight line from the uncompensated position Ⓢ to the compensated position ①' of contour point ①. The electrode center is then located perpendicular to the start of the first radius-compensated contour element.



Departure

At a transition from RL/RR to R0, the control positions the electrode center in the last radius compensated block (RL) perpendicular to the end of the last contour section.

Then the next uncompensated position is approached with R0.

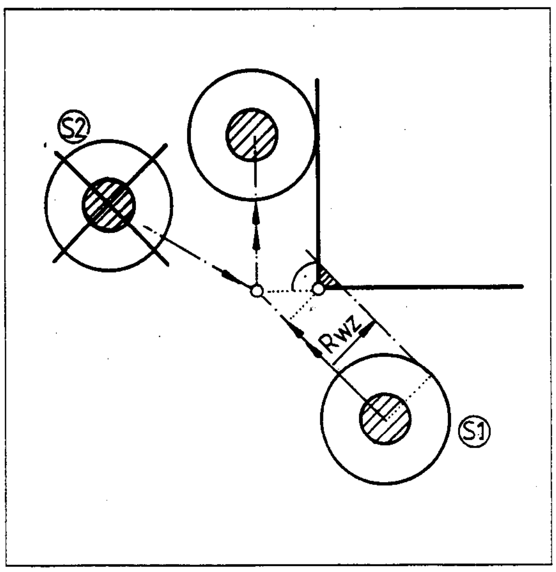


Approaching from an unsuitable position

If radius compensation is begun from S1, the electrode will damage the contour at the first contour point if no extra measures are taken!

Departure

The same applies when departing from the contour.



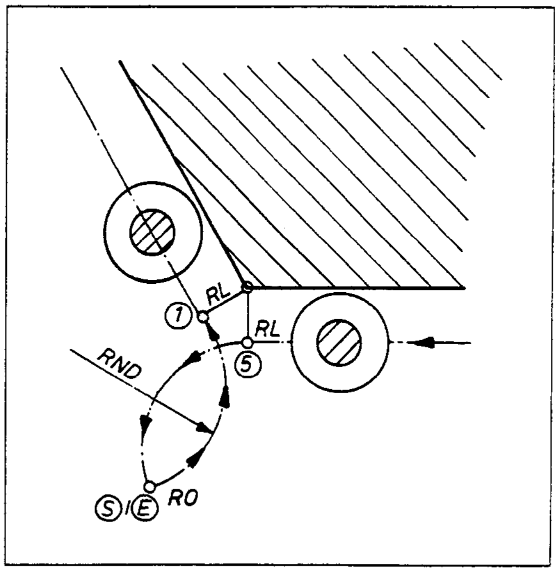
Contour Approach and Departure on an arc

Approach and departure on an arc

RND

The TNC enables you to automatically approach and depart from contours on a circular path.

Begin programming with the "RND" key.



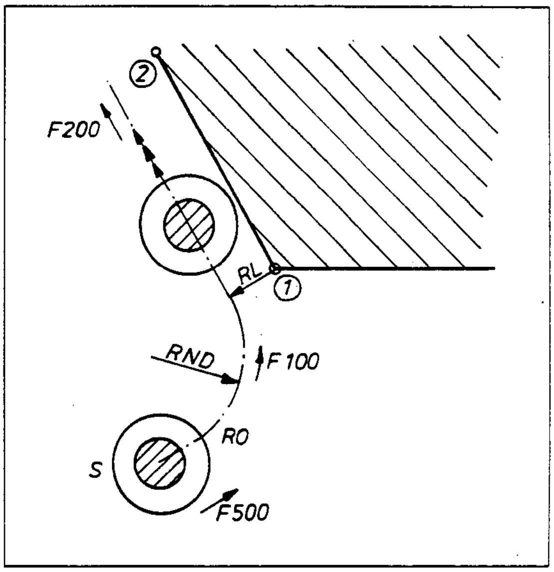
Approach

The electrode traverses from the starting position Ⓢ initially on a straight line and then on a tangentially connected arc to the programmed contour.

The starting point can be selected as desired, and is approached without radius compensation (with R0).

The straight line positioning block to contour point ① must contain radius compensation (RL or RR).

Then program a RND block.



Departure

The electrode travels from the last contour point ⑤ on a tangentially connecting arc and then on a tangentially connecting straight line to the end position Ⓔ.

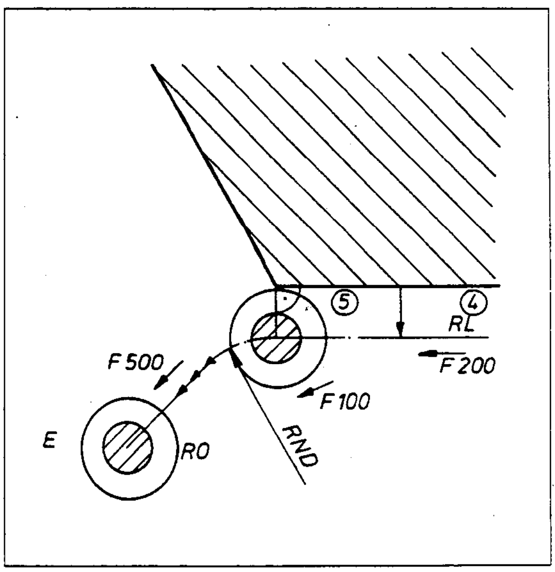
The positioning block for Ⓔ should not contain radius compensation (i. e. R0).

Approach arc/ departure arc

The radius R can be substantially less than the electrode radius. It must be small enough to fit between Ⓢ and ① or ⑤ and Ⓔ.

Program scheme

L X_S Y_S Z_S **R0**	⋮
L X_1 Y_1 **RL**	L X_5 Y_5 **RL**
RND 2.5	**RND 2.5**
L X_2 Y_2	L X_E Y_E **R0**
⋮	Z200



Notes

A positioning block containing both coordinates of the machining plane must be programmed before and after the RND block.

Approach on an arc:

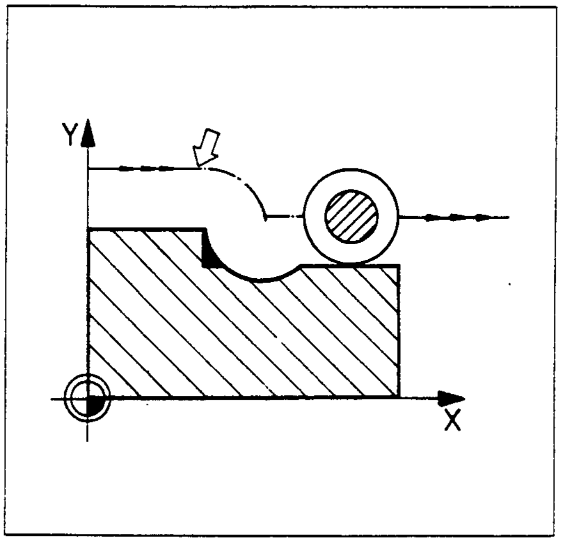
Program a RND block after the first radius compensated position (RL/RR).

Departure on an arc:

Program a RND block after the last radius compensated position (RL/RR), or before the first uncompensated position following machining.

Small contour steps: M97

If there is a step in the contour which is **smaller** than the electrode radius, the standard transition arc would cause contour damage. The control therefore issues an error message and does not execute the corresponding positioning block.



Without M97

M97

M97 prevents insertion of the transition arc. The control then determines a contour intersection Ⓢ as at inside corners and guides the electrode over this point. The contour is not damaged.

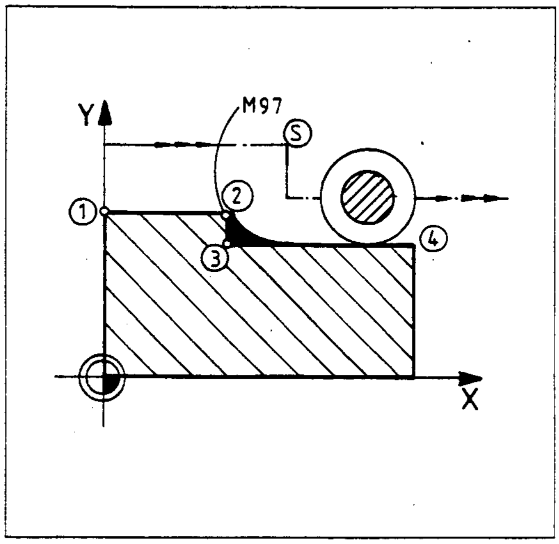
However, machining is then incomplete and the corner may have to be reworked. A smaller tool may help.

M97 is effective blockwise and must be programmed in the block containing the outside corner point.

Example

```
TOOL DEF 1 L+0 R9.5
TOOL CALL 1 Z U+0.5

L X+0 Y+30 RL M36     ①
L X+40 Y+30 M97       ②
L X+40 Y+28           ③
L X+80 Y+28           ④
```

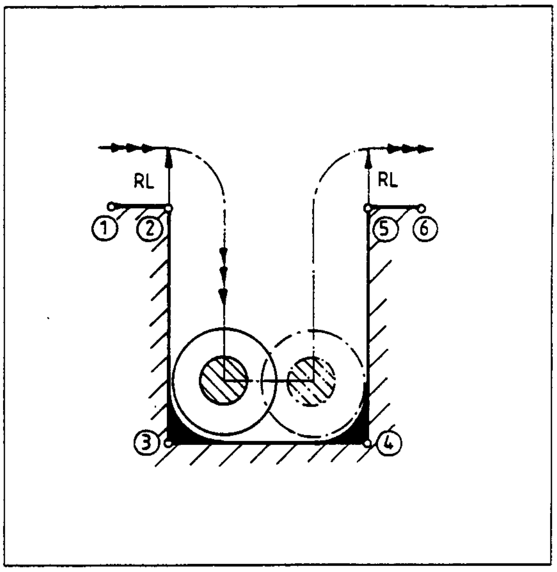


With M97

End of compensation: M98

Standard inside corner compensation

On inside corners in a continuously radius-compensated contour, the electrode moves only to the intersection of the equidistants (see top figure). The work cannot be completely machined at positions ③ and ④.



M98

The middle figure shows two independent workpieces. Positions ③ and ④ are not connected. The electrode must therefore be guided to positions ③' and ④'.

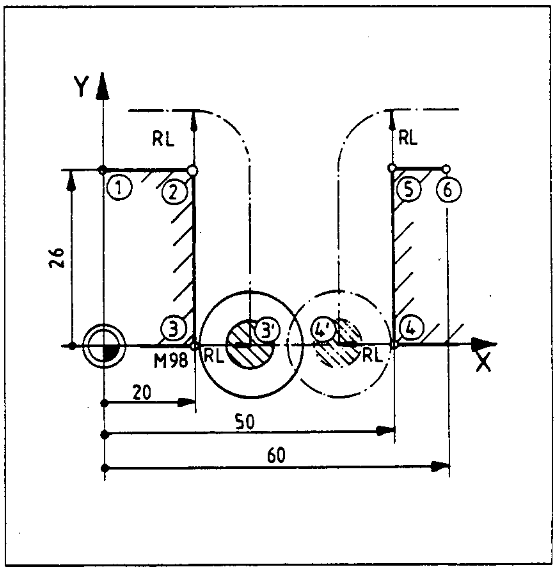
If you program a position with M98, the path offset remains valid until the end of this element and is ended there **for this block.**

No intersection is computed and **no transition arc is generated** for the end position, so the electrode is always traversed perpendicular to the contour end point.

The previous compensation is reactivated automatically in the following block ④.

Position ④' is approached perpendicularly to ④.

The contour is thus completely machined at ③ and ④.



Example

L X+0 Y+26 RL	①
L X+20 Y+26	②
L X+20 Y+0 M98	③
L X+50 Y+0	④
L X+50 Y+26	⑤
L X60 Y26	⑥

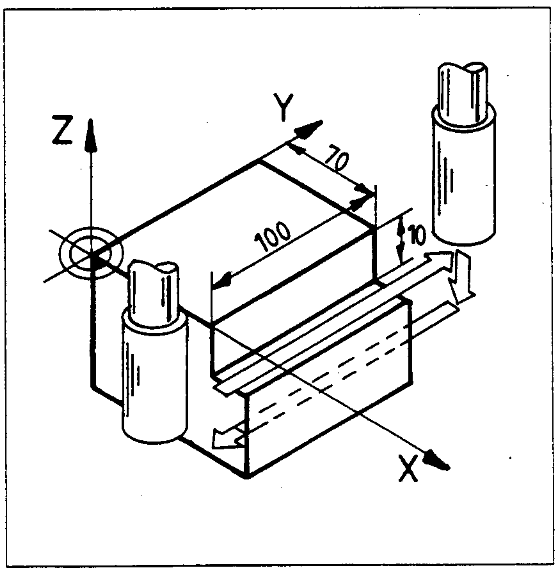
Stepover milling with M98

Stepover milling with infeeds in Z.

Example

TOOL DEF 1 L+0 R4.5
TOOL CALL 1 Z U+0.5

L X+70 Y–10 RR FMAX	Pre-positioning
L Z–10 M36	Plunge
L Y+110 **M98**	Stepover
L Z–20	Second infeed
L Y+110 RL	Pre-positioning
L Y–10 **M98**	Stepover



Predetermined M Functions
Machine-based coordinates: M91/M92

Coordinates programmed with M91 and M92 are independent of the manually set workpiece datum.

M91

Positions programmed with M91 are referenced to the datum of the linear or angle encoders. The datum is located at the negative end of the measuring range on linear encoders with distance-coded reference marks. On encoders with a single reference mark, the datum is set by this reference mark (the position of the reference mark is indicated by the **RM** sticker).

M92

When programming M92, nominal positions refer to the machine datum.

Applications

The miscellaneous functions M91 and M92 are used, for example, to

- traverse to fixed machine points, or
- traverse to the tool change position.

Displaying fixed machine coordinates

You can use the "MOD" key to display the coordinates referenced to the machine datum (see index "General Information", MOD Functions).

LBL SET LBL CALL PGM CALL

Program Jumps Overview

Jumping within a program

The following jumps can be made within a program:

- **Program section repeat**
- **Subprogram call**
- **Conditional jump**
- **Unconditional jump**

Nesting:
A program section repeat or a subprogram can also be called from within another program section repeat or subprogram.
(Maximum nesting depth: 8 levels)

Examples:

```
CALL LBL 4 REP 3/3
CALL LBL 7
IF Q5 GT0 GOTO LBL 12
IF 0 EQU 0 GOTO LBL 8
```

Jumping to another program

You can jump from a part program to any other program which is stored in the control.

Program a jump to another program with a

- **program call**

  or with

- **cycle 12: PGM CALL**

Nesting:
You can call further programs from a called program.
(Maximum nesting depth: 4 levels)

Examples:

```
CALL PGM 3
CYCL DEF PGM CALL PGM 3
.
.
.
L X+50 M99
```

LBL SET LBL CALL

Jumping Within a Program
Program markers (labels)

Labels

Labels (program markers) can be set during programming to mark the beginning of a subprogram or program section repeat.

These labels can be jumped to during program run (e.g. to execute the appropriate subprogram).

Setting a label

LBL SET

A label is set with the "LBL SET" key. The label numbers 1 to 254 can be set only once in a program.

Label 0

Label number 0 always marks the end of a subprogram (see "Subprogram") and is therefore the return jump marker. It can thus occur more than once in a program.

Calling a label number

LBL CALL

The dialog is initiated with the "LBL CALL" key.

With LBL CALL you can:

- call subprograms
- create program section repeats.

Label numbers (1 to 254) can be called as often as desired.
Do not call label 0!

Program section repeats

For program section repeats, respond to the query REPEAT REP ? by entering the number of required repetitions.

Subprograms

For subprogram calls, respond to this query with the "NO ENT" key.

Conditional jumps

You can make the call of a program label be dependent on a mathematical condition (see Parametric Programming, Overview).

Error messages

JUMP TO LABEL 0 NOT PERMITTED
This jump (CALL LBL 0) is not allowed.

LABEL NUMBER ALLOCATED
Each label number – except LBL 0 – can be allocated (set) **only once** in a given program.

Jumping Within a Program
Program section repeats

Program section repeats

An executed program section can be executed again immediately. This is called a program loop or program section repeat.

LBL SET

A label number marks the beginning of the program section which is to be repeated.

LBL CALL REP with number

The end of the program section to be repeated is designated by a call LBL CALL with the number of repetitions REP.

A program section can be repeated up to 65534 times.

Jump direction

A called program section repeat is always executed completely, i.e. until LBL CALL.
A program jump is therefore only meaningful if it is a return jump. In other words, the called label (LBL SET) must have a smaller block number than the calling block (LBL CALL).

```
.
.
.
22 LBL 2
23 L IX+10   FMAX M99
24 CALL LBL 2 REP 5 /5
.
.
.
```

Program run

The control executes the main program (along with the associated program section) until the label number is called. Then the return jump is carried out to the called program label and the program section is repeated.

The number of remaining repetitions on the display is reduced by 1: REP 2/1.

After another return jump, the program section is repeated a second time.

When all programmed repetitions have been performed (display: REP 2/0), the main program is resumed.

The total number of times a program section is executed is always one more than the programmed number of repeats.

```
.
.
.
22 LBL 2

23 L IX+10
   FMAX M99

24 CALL LBL 2
   REP 5 /5
.
.
.
```

Error message

EXCESSIVE SUBPROGRAMMING

A jump was programmed incorrectly:

1. No REP value was entered for a program section repeat.
   If no response is given to the query REP (by pressing the "NO ENT" key), the program section is treated like a subprogram without a correct ending (LBL 0): the label number is called 8 times. During program run or a test run, the error message appears on the screen after the 8th repetition.
2. The subprogram was programmed without LBL 0 for an intended subprogram call.

Jumping Within a Program
Program section repeats

Setting the program label

Example:

LBL SET 1 ENT

Program label 1 is set.

:

Repeating a program section after LBL

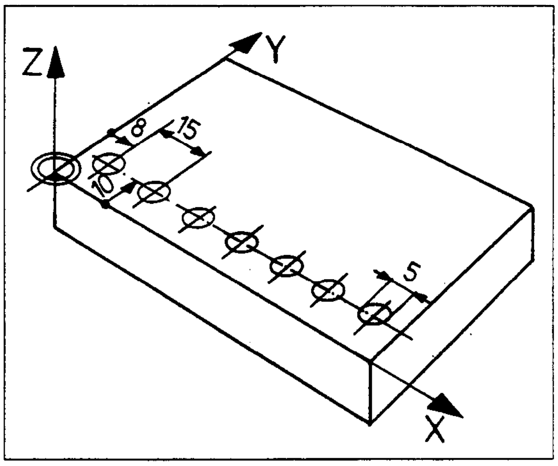
LBL CALL 1 ENT 6 ENT

6 repetitions after LBL 1.
The program section between LBL 1 and CALL LBL 1 is executed a total of 7 times.

Example bolt-hole row

The illustrated bolt-hole row with 7 identical holes is to be eroded with a program section repeat.

The tool is pilot positioned (offset to the left by the hole center distance) before starting the repeat to simplify programming.



Program

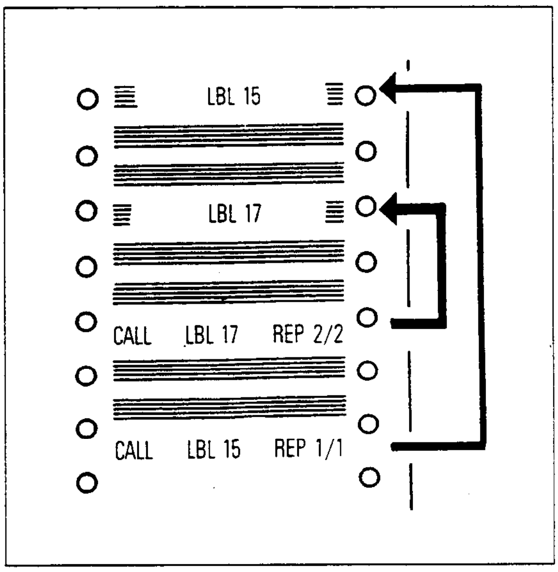
TOOL DEF 1 L+0 R2.4	Tool definition
TOOL CALL 1 Z U+0.1	Tool call
L X–7 Y+10 Z+2 R0 FMAX M3	Pre-positioning
LBL 1	Start of the program section repeat
L IX+15 FMAX	Incremental distance between the holes, rapid traverse
L Z–10 M36	Absolute sinking depth, sinking feed rate
L Z+2 FMAX M37	Absolute retraction height, rapid traverse
CALL LBL 1 REP 6	Call for repeats

Nesting of repetitions

The main program is executed until the jump to LBL 17 (CALL LBL 17).
The program section between LBL 17 and CALL LBL 17 is repeated twice.

The control then resumes the main program run until the jump to LBL 15 (CALL LBL 15).

The program section until CALL LBL 17 REP 2/2 is repeated once and the nested program section also two more times. Then the program run is resumed.



Jumping Within a Program
Subprograms

Subprograms

If a program section occurs several times in the same program, it can be designated as a subprogram and called whenever required. This speeds up programming.

Start of subprogram

The start of subprogram is marked with a **label number** (can be any number).

End of subprogram

The end of the subprogram is always marked by **label 0.**

The different subprograms are then called in the main program as often as wanted and in any sequence.

```
.
.
.
14 CALL LBL 1
15 L X+20 Y+50
16 CALL LBL 1
17 L X+10 Y+80
18 CALL LBL 1
19 L Z+50 R0 M02

20 LBL 1
21 CYCL CALL M
22 LBL 2
23 L IX+10 M99
24 CALL LBL 2 REP 5 /5
25 LBL 0

26 END PGM 1 MM
```

No repetitions Reply to REPEAT REP with NO ENT

When the subprogram is called with LBL CALL, the "NO ENT" key must be pressed after the dialog query REPEAT REP ? appears. A subprogram can be called at any point in the main program (but not from within the same subprogram).

Program run

The control executes the main program until the subprogram call ①.

A jump to the called program label ② is then performed.

Subprogram 1 is processed until label 0 (③) (end of subprogram).

Then the return jump to the main program follows. The main program is resumed with the block ④ following the subprogram call.

```
.
.
.
① CALL LBL 1
④ L X . . . Y . . .
.
.
.
.
                    M02

② LBL 1
.
.
.
③ LBL 0
```

Subprograms should be placed after the main program (behind M2 or M30) for the sake of clarity. If a subprogram is placed within the main program, it is also executed once during program run without being called.

Error messages

If a subprogram call is programmed incorrectly (e.g. an end of subprogram lacks LBL 0, or a value for REPEAT REP ? was entered), the error message

EXCESSIVE SUBPROGRAMMING

appears.

LBL CALL

Jumping Within a Program
Subprograms

Entry example: Subprogam 2

BEGIN PGM 1 MM
:
:

LBL CALL 2 ENT NO ENT — Subprogram 2 is called from within the main program. Conclude with "NO ENT"

.
.
.

L Z100 FMAX M2 — Retract and return jump to start

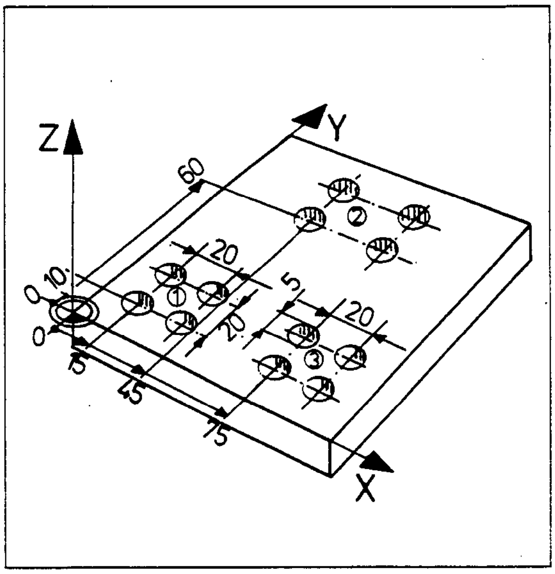
LBL 2 — Start of subprogram 2
.
.
.

LBL 0 — End of subprogram 2

END PGM 1 MM — End of main program

Example

A group of four holes is to be programmed as subprogram 1 and executed at three different positions.



Program

TOOL DEF 1 L+0 R2.4	
TOOL CALL 1 Z U+0.2	
L X+15 Y+10 R0 FMAX	Traverse to hole group ①
CALL LBL 1	Subprogram call
L X+75 Y+10 FMAX	Traverse to hole group ②
CALL LBL 1	Subprogram call
L X+45 Y+60 FMAX	Traverse to hole group ③
CALL LBL 1	
L Z+50 FMAX M2	Retract tool axis
LBL 1	Subprogram 1
CALL LBL 2	Subprogram call for sinking and retracting
L IX+20 FMAX	Incremental traverse
CALL LBL 2	Sinking
L IY+20 FMAX	Incremental traverse
CALL LBL 2	Sinking
L IX–20 FMAX	Incremental traverse
CALL LBL 2	Sinking
LBL 0	Subprogram 1 end
LBL 2	Subprogram 2
L Z–10 M36	Sink
L Z+2 FMAX M37	Retract
LBL 0	End of subprogram 2

Program Calls

Jumping to another main program

You can call another program which is stored in the control from any machining program.
This allows you to create your own fixed cycles with parametric programming.
Program the call with a "PGM CALL" key.

Calling criteria

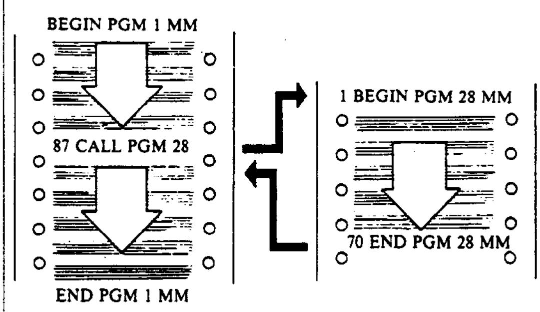
The program to be called cannot contain M02 or M30. In the called program, do not program a jump back to the original program (creates an endless loop). Only one BLK FORM can exist.

Process

The control executes main program 1 until CALL PGM 28. Then a jump is made to main program 28.

Main program 28 is executed from beginning to end.

Then a return jump is made to main program 1. Main program 1 is resumed with the block following the program call.



Example 1

PGM CALL 10 ENT

CALL PGM 10	Call with a separate program line

Example 2

The program to be called can also be specified with a cycle definition. The call then functions like a fixed cycle.

CYCL DEF GO TO 1 2 ENT

CYCL DEF 12.0 PGM CALL CYCL DEF 12.1 PGM 20	Call e. g. via M99 (see Other Cycles, Cycle 12)

Standard Cycles
Introduction, Overview

Standard cycles

To facilitate programming, frequently recurring machining sequences (orbital functions), specific erosion functions, certain coordinate transformations and other programming aids are programmed as standard cycles.

OEM cycles

The machine manufacturer can also store his own programs as cycles in the control. These cycles can be called under the cycle numbers 68 to 99. Contact the machine manufacturer for more information.

No.	Cycle	Effective upon call	Immediately effective
1	Generator		●
17	Disk		●
2	Erosion with time limit		●
3	Tool definition		●
7	Datum shift		●
8	Mirror image		●
10	Rotation of coordinate system		●
11	Scaling		●
9	Dwell time		●
2	Program call	●	

Selecting a cycle

CYCL DEF

After pressing the "CYCLE DEF" key, data for the cycles shown to the right can be entered and also any programmed user cycles can be selected. The desired cycle can be selected with the vertical cursor keys or with "GOTO □".

Defining a cycle

The cycle definitions can be entered in the dialog after pressing "ENT".

Calling a cycle

Cycles must be called after moving the tool to the appropriate position – only then will the last defined cycle be executed.

There are three ways to call a cycle:

CYCL CALL

- With a separate CYCL CALL block

M99

- Via the miscellaneous function M99. "CYCL CALL" and M99 are only effective blockwise and must therefore be reprogrammed for every execution.

M89

- Via the miscellaneous function M89 (depending on machine parameters).

M89 is effective modally, i. e. the last programmed cycle is called at every subsequent positioning block. M89 is cancelled or cleared by M99 or by CYCL CALL.

Coordinate transformations

Coordinate transformations and the dwell time are effective immediately and remain effective until changed.

Erosion Cycles
Cycle 1: Generator

Input data

Erosion table P-TAB:
Number by which the required erosion parameter table can be called (with M36, erosion on).

Highest and lowest power stage MAX and MIN:
Numbers of the highest and lowest power stages needed for the following machining task.

Description

The generator cycle selects the erosion parameters of the highest defined power stage to begin the following machining task. With the parameter Q99 the power stage can be changed within the defined range.

If you are using erosion tables, you must program the generator cycle before beginning the erosion process (M36). The cycle is not needed if erosion tables are not being used. After the generator cycle is run in the "Program run/single block" and "Program run/full sequence" operating modes the highest and lowest defined power stages are shown beneath the position display (e.g. NR 24–10).

If you are not using erosion parameter tables, be sure to regard the Q parameters Q90 to Q99.
If you are using tables, the Q-parameters Q96 to Q255 apply (see also the section "Parametric Programming").

Electrode undersize

Operating mode

Initiate the dialog CYCL DEF

CYCL DEF 1 GENERATOR	ENT Confirm selection.

ERODING TABLE?	Number of the required erosion parameter table. ENT Confirm entry.

POWER STAGE MAX?	Highest power stage. ENT Confirm entry.
POWER STAGE MIN?	Lowest power stage. ENT Confirm entry.

Introduction

Unlike milling and other conventional machining methods, spark erosion is strongly influenced by many process variables, called erosion parameters. The erosion parameter values must change depending on the combination of tool and work-piece materials or the type of machining, for example roughing or finishing. Parameter tables simplify your work by grouping these diverse values according to specific tasks at hand.

In the generator cycles, the control either transmits these parameter values to the generator (LV, HV, GV, T-ON as well as AR, P and HS), or it processes them internally (SV, AJD and ET).

Other parameter values are needed for calculations (2G and UNS), as geometric data (e.g. expansion radius RAD in the disk cycle), or to influence the selection of the appropriate power stage (WR, RA and SR). The auxiliary parameters (AUX 1 to AUX 6) are defined by the machine tool builder.

The following parameter descriptions may differ slightly from their functions on your specific machine tool.

```
PARAMETER TABLE 300
POWER STAGE NUMBER
    TAB:300                        NR:25   >>
NR  LV  HV  GV  T-ON   TF  SV  AJD   ET
25  99  9   99  100    10  99  1,5   5
24  80  8   85  90     10  99  1,5   5
23  76  8   72  90     20  99  1,5   5
22  70  7   70  80     20  99  1,5   5
21  60  7   60  80     30  99  1,5   5
20  50  7   45  80     40  99  1,5   5
-----------------------------------------
ACTL. X       -0,188   Y       -0,199
      Z       -0,185   C     +359,808


                              F
```

Overview

The erosion parameters and their meanings:

Abbreviation	Function	Input range	Unit
NR	Power stage	25 – 1	
LV HV GV T-ON TF	Low Voltage current High Voltage current Gap Voltage Pulse-on duration Pulse-off duration	0 - 99 0 - 9 0 - 99 0 - 999 0 - 99	
SV	Servo sensitivity	0 - 99	%
AJD ET	Auto Jump Distance Erosion Time	0 - 99.9 0 - 999	mm sec.
AR P HS	Arc sensitivity Electrode polarity: 0 = +, 1 = – High voltage Selector	1 - 99 0/1 0 - 99	
WR RA SR	Wear Rate Surface finish Stock removal	0 - 99 0 - 99.9 0 - 999.999	% µm ccm/min
2G UNS	Two-times Gap Minimum undersize	0 - 9.999 0 - 9.999	mm mm
AUX 1 AUX 2 AUX 3 AUX 4 AUX 5 AUX 6	Auxiliary parameters AUX 1 to AUX 6 are determined by the machine tool builder	0 - 99 0 - 99 0 - 99 0 - 99 0 - 9999 0 - 9999	

NR

Power stage

Each erosion parameter table can store up to 25 power stages. The power stages determine the type of machining. High power stages are needed for roughing, mid-range power for finishing and low power for fine finishing or polishing.

The power stages must be programmed in decreasing sequence, with an increment of 1.

The required power stage can be selected by the Q parameter Q99 (see also the section "Parametric Programming").

LV

Low voltage current

At low erosion voltages, the current can be programmed in up to 100 stages.
Your machine tool builder can provide you with more information on this parameter.

HV

High voltage current

At high erosion voltages the current can be programmed in up to 10 stages.
Your machine tool builder can provide you with more information on this parameter.

GV

Gap voltage

In order to achieve optimum results, the erosion gap must be set according to the machining task. The correct gap is achieved with the aid of the gap nominal value GV and is maintained throughout the erosion process by the gap control.

Small gap voltages result in a high stock removal rate. An excessively small GV, however, can result in process defects such as short circuiting and arcing.

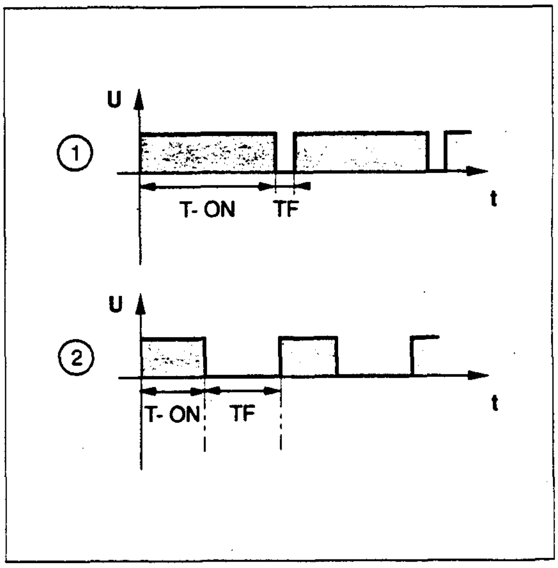
T-ON and TF

Pulse-on duration, Pulse-off duration

The pulse-on duration T-ON defines the time in which the generator is switched on for a spark with subsequent electrical discharge.

The pulse-off duration TF defines the time in which the generator is off. During this time the gap is flushed and deionized.

The ratio of pulse-on to pulse-off time, also called the pulse duty cycle, determines the type of machining. A high duty cycle is needed for roughing ①, while a low duty cycle is required for finishing or fine finishing ②.



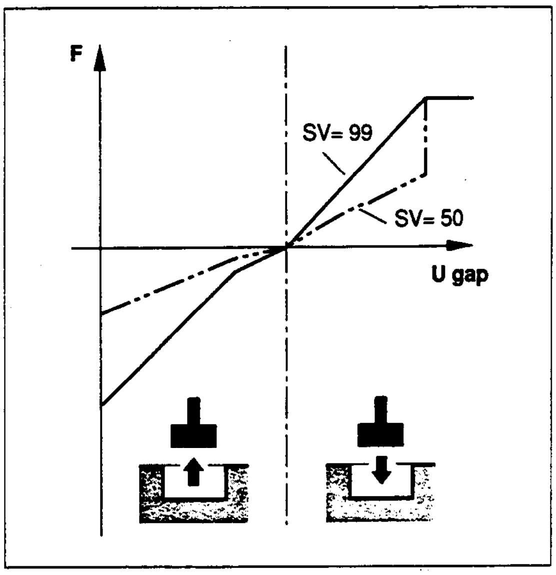
SV

Servo sensitivity

The control outputs a certain velocity nominal value F depending on the gap voltage U_{gap} and the characteristic curve adjusted by the machine tool builder.

The reaction speed of the gap control can be influenced with the servo sensitivity parameter SV.

Servo sensitivity is entered in %.



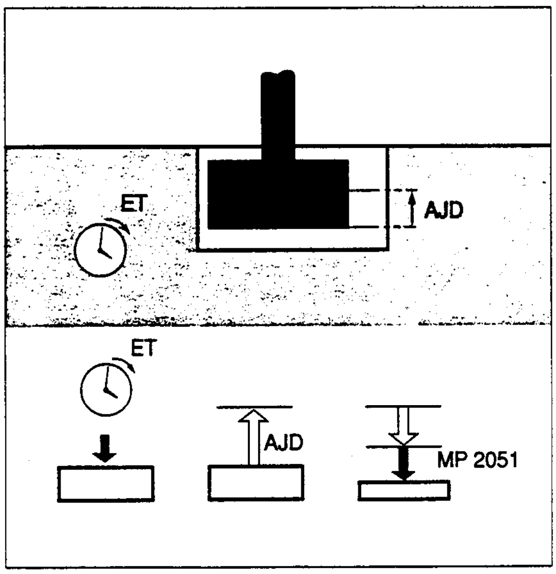
AJD and ET

Auto jump distance, Erosion time

The erosion process is completed within the erosion time ET. When the erosion time expires, the electrode is quickly retracted along the programmed path by the auto jump distance AJD. Then the electrode is quickly returned to the contour, abbreviated by MP2051.

An additional, intermittent flushing function can be switched on to prevent process malfunctions and improve deionization of the gap. The corresponding M function is assigned by the machine tool builder, e.g. M8.

The auto jump distance AJD is entered in millimeters and the erosion time ET in seconds.



⬇ = Erosion
⇩ = Positioning

AR

Arc sensitivity

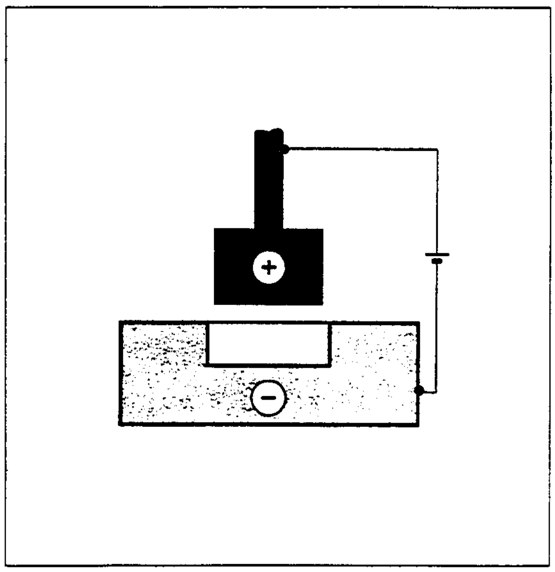
The arc sensitivity AR influences the gap signal sent from the generator to the TNC. Your machine tool builder can provide you with more information on this parameter.

P

Electrode polarity

The polarity of the electrode must be selected to fit the combination of tool and workpiece material. This parameter has an important influence on stock removal and wear rate.

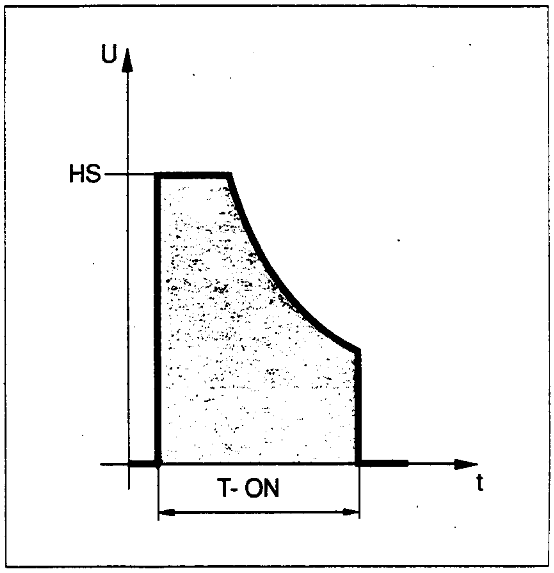
- Positive electrode = 0
- Negative electrode = 1



HS

High voltage selector

During the pulse-on time, the generator produces a certain voltage between the electrode and the workpiece before the electrical discharge. This voltage corresponds to the high voltage selector.

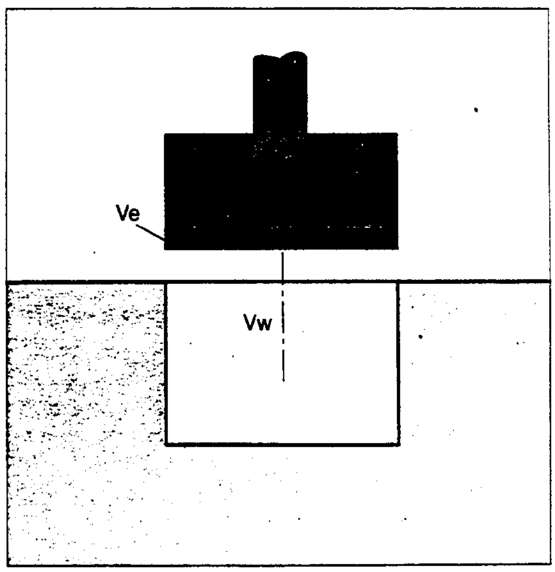


WR

Wear rate

The wear rate is the ratio of material volume removal from the electrode V_e (electrode wear volume) and from the workpiece (V_W), expressed in %.

$$WR = \frac{V_e}{V_W} \cdot 100$$



RA

Surface finish

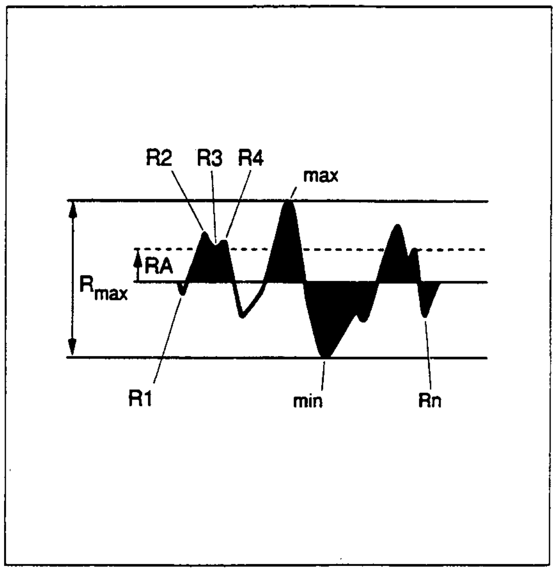
If you look at the surface of a contour through a powerful magnifying glass, you will see peaks and valleys. The difference in height between the highest peaks and the lowest valleys is the maximum surface roughness R_{max}.
All peaks and valleys are measured from the mean line of the profile. The sum of the points R divided by the number n of all points results in the average for the surface roughness RA.

$$RA = \frac{R1 + R2 + \ldots + Rn}{n}$$

and

$$Rmax = \frac{UNS - 2\,G}{2}$$

The surface finish RA is given in µm.

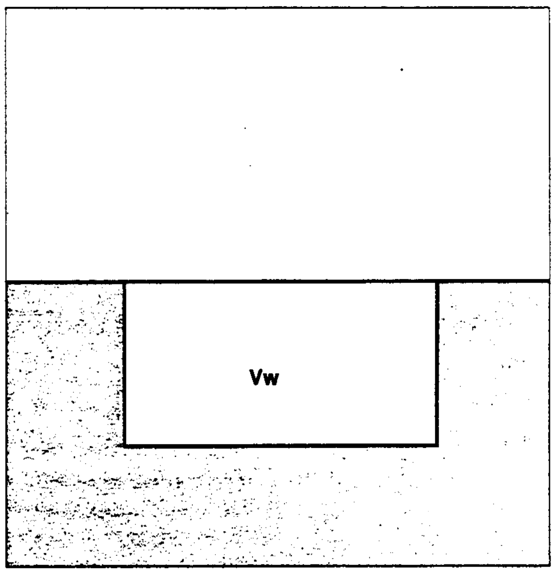


SR

Stock removal

The volume of workpiece material V_W is called the stock removal. This quantity is proportional to the average erosion speed.

The stock removal SR is given in $\frac{cm^3}{min}$
($1\ cm^3 = 1000\ mm^3$).



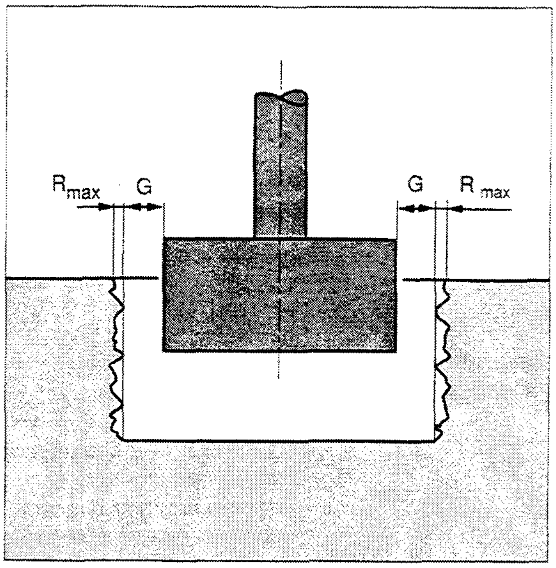
2G

Two-times gap

The size of the gap G depends more than anything else on the adjusted current level. Since the gap G exists on both sides of the electrode, it must be counted twice. This parameter is therefore referred to as the two-times gap 2G.

$$2\,G = G + G$$

The two-times gap is given in mm.



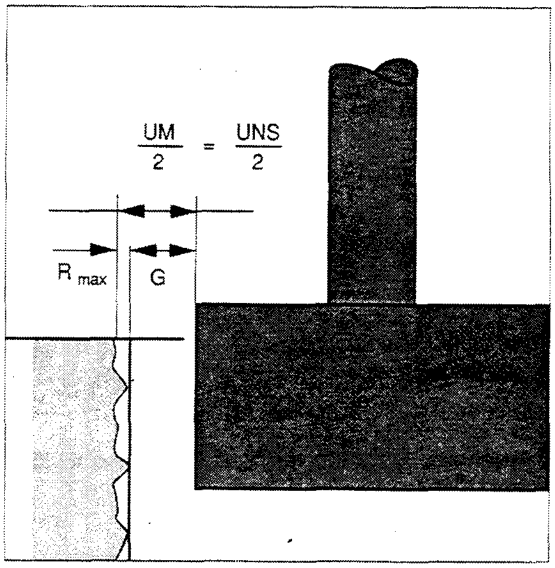
UNS

Minimum undersize

The minimum undersize UNS results from the width of the gap G and the maximum surface roughness R_{max} (negligible for finishing).
In a simple erosion job the minimum undersize UNS is equal to the undersize UM. If you are working with the disk cycle, however, the undersize UM is greater than the minimum undersize UNS.

$$UNS = 2\,G + 2\,Rmax$$

$$UM \geqq UNS$$



AUX 1 to AUX 6

Auxiliary parameters

The auxiliary parameters AUX 1 to AUX 6 can be used by the machine tool builder for special tasks. He can provide you with more information about their specific functions.

Erosion Cycles
Cycle 17: Disk

Introduction

The disk cycle is a general-purpose cycle intended to be used for the development of user-specific erosion cycles.
With Q parameters and the disk cycle you can program, for example, conical or spherical cavities (see section "Parametric Programming").

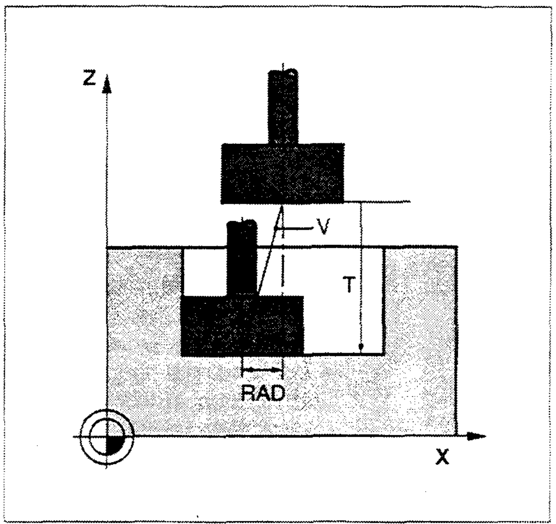
Prerequisite

The disk cycle is effective as soon as it is defined. The following items must therefore be programmed **before** calling and defining the disk cycle:

- Tool call (tool axis, tool compensation, undersize)
- Positioning to the start position S (disk center)

Erosion axis and depth

Entering the erosion axis (e.g. Z) identifies the axis for the programmed total cavity depth. The working direction of erosion is defined by the algebraic sign of the entered value.
\+ Positive working direction
– Negative working direction
The depth can be entered as either an **absolute** or **incremental** value.



Miscellaneous function M

The disk cycle accepts miscellaneous functions (e.g. M36 erosion on).

Expansion radius RAD

The expansion radius RAD is the distance by which the electrode is fed away from the disk center.

The electrode radius R_e must be greater than the expansion radius RAD. Otherwise the pocket will be incompletely machined.

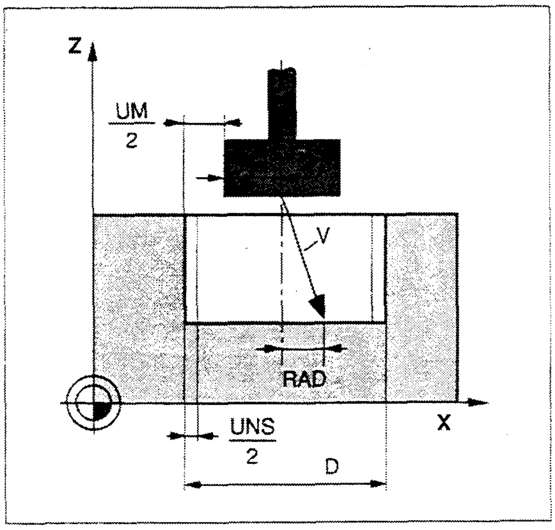
Formula:

$$RAD = \frac{UM - UNS}{2}$$

since $UM = D - 2R_e$, it follows that:

$$RAD = \frac{D}{2} - R_e - \frac{UNS}{2}$$

RAD = Expansion radius
UM = Electrode undersize
UNS = Minimum undersize
D = Disk diameter
R_e = Electrode radius



Erosion Cycles
Cycle 17: Disk

Mode

The value (between 0 and 7) programmed in the cycle parameter MOD influences both the geometric and electrical processes.

Fast sparking out

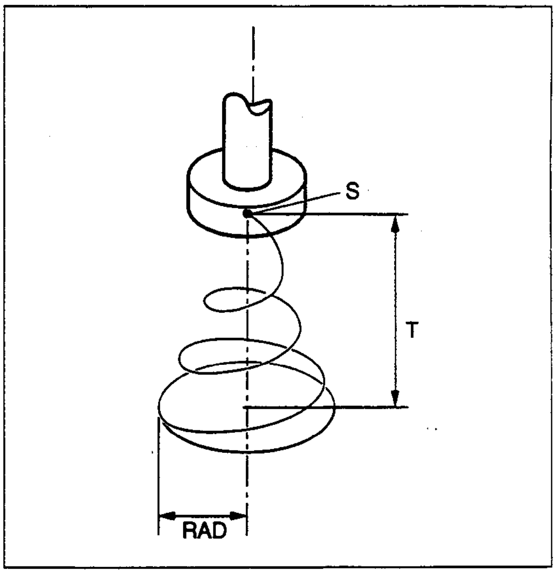
Modes 0 to 2:
The values 0 to 2 define various geometric processes. After the electrode reaches the final vector V – consisting of total depth and total radius – and completes one full orbit, the cycle ends.

Complete sparking out

Modes 4 to 6:
Values 4 to 6 define the same geometric processes as the values 0 to 2. After the electrode reaches the final vector V, the generator transmits the sparking out signal (free-running signal) for the duration of 1¼ orbits to ensure complete sparking out before the cycle ends.

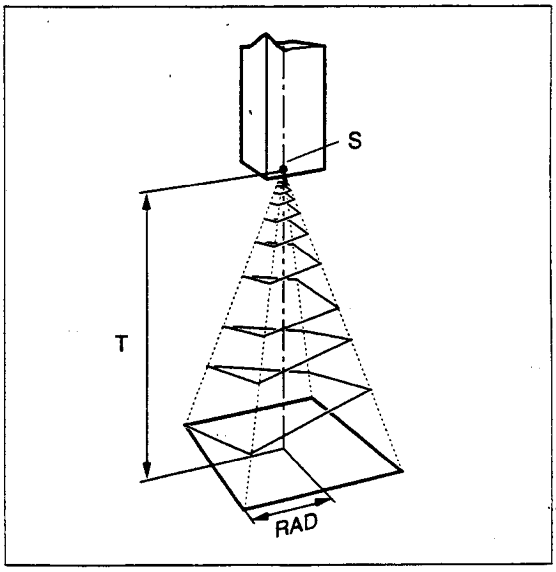
Circular expansion

Modes 0 and 4:
In the modes 0 and 4 the electrode moves from the starting point S along the surface of a circular cone until it reaches the programmed depth T and the programmed expansion radius RAD.



Square expansion

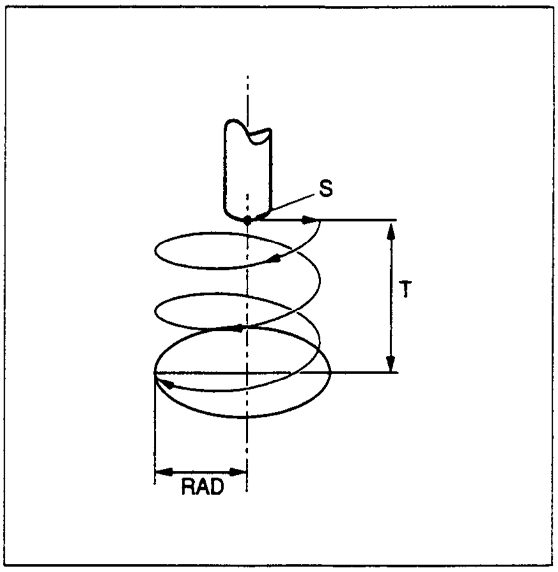
Modes 1 and 5:
In the modes 1 and 5 the electrode moves from the starting point S along the surface of a square pyramid until it reaches the programmed depth D and the programmed expansion radius RAD.



Erosion Cycles
Cycle 17: Disk

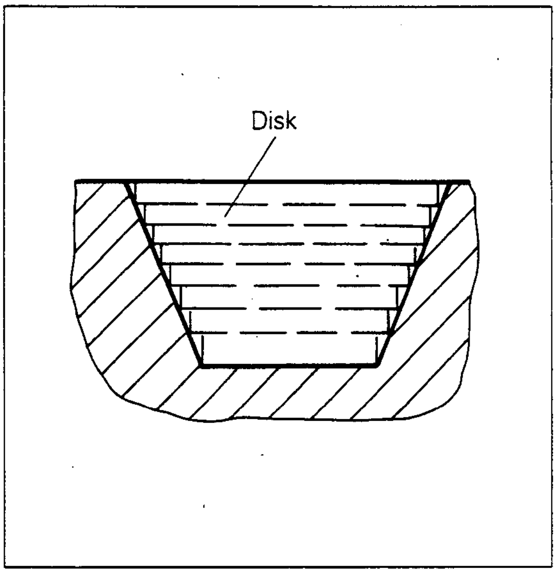
Orbital sinking

Modes 2 and 6:
In the modes 2 and 6 the electrode moves from the starting point S by the expansion radius RAD in radial direction, then takes a helical path to the programmed depth T.

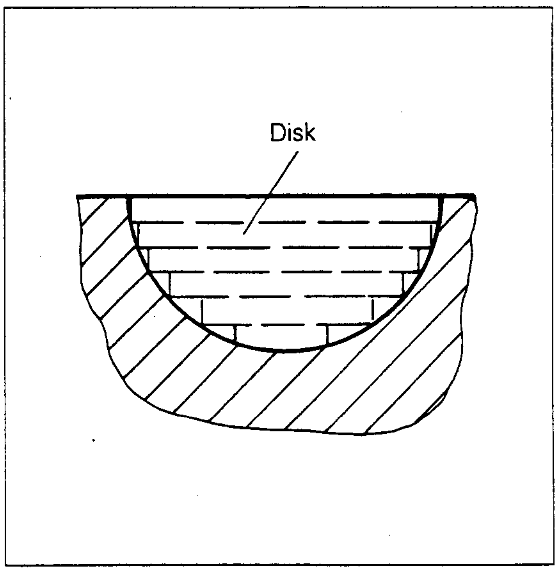


Note

- The feed rate for the rotational traverse is the last programmed feed rate. It is limited by the user parameters MP 1092 to MP 1097.
  The feed rate in the direction of the tool axis is determined by the gap monitoring.
- In the event of a short circuit, the circular movement is stopped and the electrode is retracted along the infeed vector. Once the short circuit is eliminated, the electrode moves along the infeed vector back to the last eroded position minus the value from user parameter MP 2050.[1]



Conical cavity from several disks



Spherical cavity from several disks

[1] The retraction behavior of the electrode in the event of short circuiting is determined by the machine tool builder.

Erosion Cycles
Cycle 17: Disk

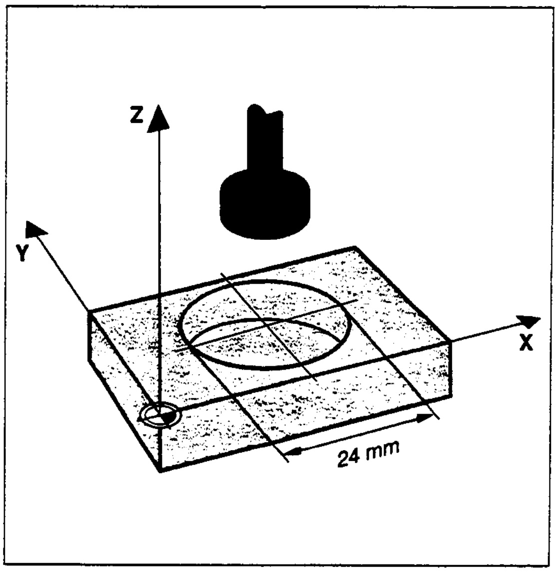
Example:

A cavity with 24 mm diameter and 10 mm depth. The cylindrical electrode has a radius of 9.9 mm and an undersize of 4.2 mm. The erosion gap is 0.1 mm in width.

The following calculation results in an expansion radius of 2 mm for the disk cycle:

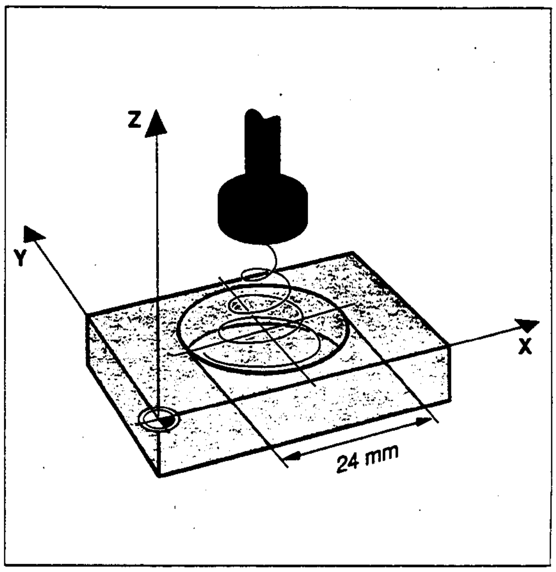
$$RAD = \frac{4.2\ mm}{2} - 0.1\ mm = 2\ mm$$

In the first program example the electrode should be above the surface of the workpiece before cycle call. In program 2, however, the electrode has already eroded to a depth of 10 mm before the disk cycle is called (circular expansion).



Program 1

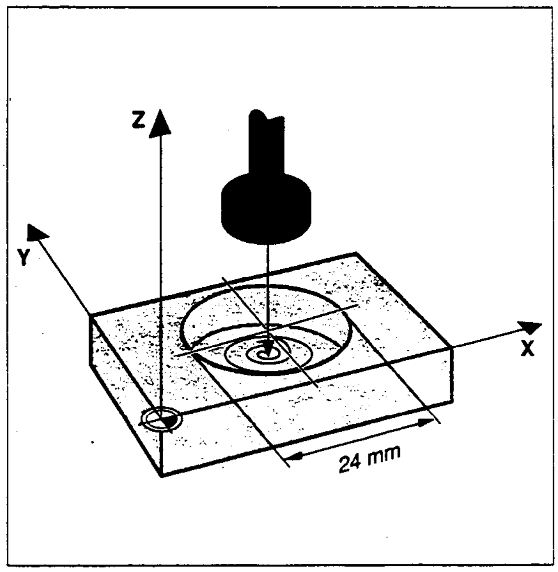
TOOL DEF 1 L+0 R9.9 TOOL CALL 1 Z U+4.2	Tool definition
L X+50 Y+50 Z+1 R0 FMAX	Traverse to start position
CYCL DEF 17.0 DISK CYCL DEF 17.1 Z–10 M36 CYCL DEF 17.2 RAD=2 MOD=0	Simultaneous erosion to depth –10 mm and expansion of the radius by 2 mm.
L X–20 Y–20 Z+100 R0 FMAX M37	Retract and terminate erosion



Path of electrode: program 1

Program 2

TOOL DEF 1 L+0 R9.9 TOOL CALL 1 Z U+4.2	Tool definition
L X+50 Y+50 Z+1 R0 FMAX	Traverse to center of cavity
L Z–10 R0 M36	Sink to start position
CYCL DEF 17.0 DISK CYCL DEF 17.1 Z+0 M CYCL DEF 17.2 RAD=2 MOD=0	Circular expansion in the depth Z = –10 mm
L Z+1 R0 FMAX M37	Retract from cavity
L X–20 Y–20 R0 FMAX	



Path of electrode: program 2

Erosion Cycles · Cycle 2: Erosion with time limit

Input data

Erosion time T:
Duration of erosion process in minutes.

Description

This cycle is used exclusively in connection with the **disk** cycle. The "erosion with time limit" cycle is defined before the disk cycle. In this cycle the duration of the disk cycle is entered in minutes. When the erosion time is over the disk presently being machined is finished (see Q parameter Q158 in the section "Parametric Programming").

Cycle definition

Operating mode

Initiate the dialog CYCL DEF

CYCL DEF 2 TIME-ERODING	ENT Select cycle.

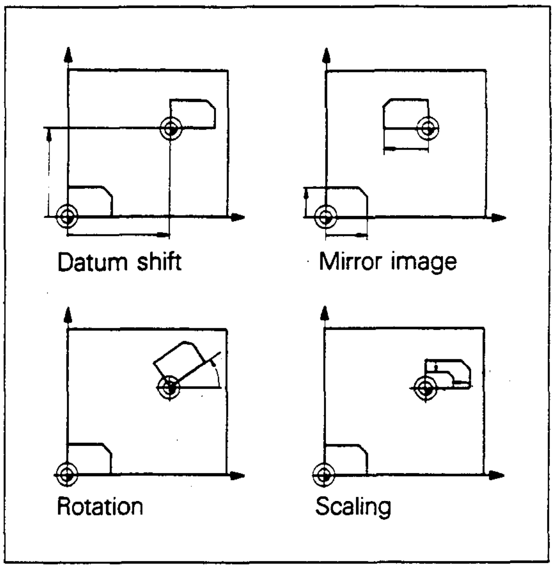
EROSION TIME IN MINUTES?	Erosion time. ENT Confirm entry.

Coordinate Transformations Overview

The following cycles serve for coordinate transformations:

3 Tool definition

7 Datum shift
8 Mirror image
10 Rotation
11 Scaling



Original

With the help of coordinate transformations, a program section can be executed as a variant of the "original".

In the following descriptions, subprogram 1 is always the "original" subprogram (identified by the gray background).

Immediate activation

Every transformation is immediately valid – without being called.

Duration of activation

A coordinate transformation remains valid until it is changed or cancelled.

Its effect is not impaired by interrupting and aborting program run. This is also true when the same program is restarted from another location with "GOTO □".

End of activation

You can cancel coordinate transformations in the following ways:

- Cycle definition for basic condition (e. g.: scaling factor 1.0);
- Programming of miscellaneous functions M02 or M30, or END PGM ... (depending on the machine parameters);
- Selecting another program with "PGM NR" in the operating mode program run "full sequence" or "single block".

Error message

CYCL INCOMPLETE

This error message is displayed if a fixed cycle is called after defining a transformation but no machining cycle was defined. Otherwise the control executes the fixed cycle which was last defined.

Coordinate Transformations
Cycle 3: Tool definition

Input data

Tool number:
Number of the tool to be defined.
Permissible tool numbers: 1 – 9999.

Tool radius:
The tool radius must be positive and is used to calculate the radius compensation.

Tool compensation:
The compensated axis and the associated positive or negative length compensation can be entered for a maximum of 4 axes (X, Y, Z and C axes).

Cycle definition

Initiate dialog — CYCL DEF ↓ or GOTO 3 ENT

CYCL DEF 3 TOOL DEF	ENT Select cycle.
TOOL NUMBER ?	Tool number. ENT Confirm entry.
TOOL RADIUS R ?	Tool radius. ENT Confirm entry.
TOOL COMPENSATION ?	X Select axis. Enter tool compensation value. Y : Tool compensation is possible in all 4 axes. END Conclude block only after values for all compensated axes have been entered.

Coordinate Transformations
Cycle 3: Tool definition

Description

Tools can be defined with the TOOL DEF function or with cycle 3 with tool compensation in up to 4 axes (optional).

Tool number and tool radius input have the same meaning as in the TOOL DEF function.

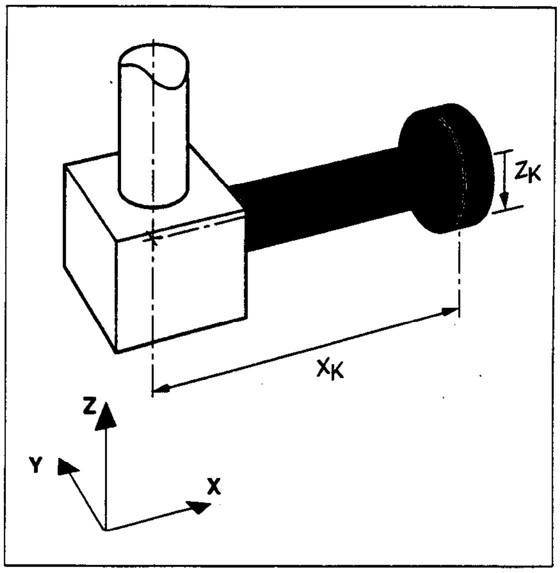
It is also possible to compensate the electrode in up to 4 axes. This compensation shifts the tool datum by the entered values.

Example:

A 5 mm hole is to be eroded at X+50 and Y+50 with the illustrated electrode.
X compensation: –10
Z compensation: –5
After tool definition you can then position the tool with a linear block to X+50 and Y+50 and begin erosion.

Program:

```
        :
CYCL DEF 3.0 TOOL DEF
CYCL DEF 3.1 T 1                    }
             R+0                    } Tool
CYCL DEF 3.2 X–10 Z–5               } definition
TOOL CALL 1 Z U+0.1                   Tool call
L X+50 Y+50 Z+0                       Positioning
L Z–5 M36                             Erosion
        :
```



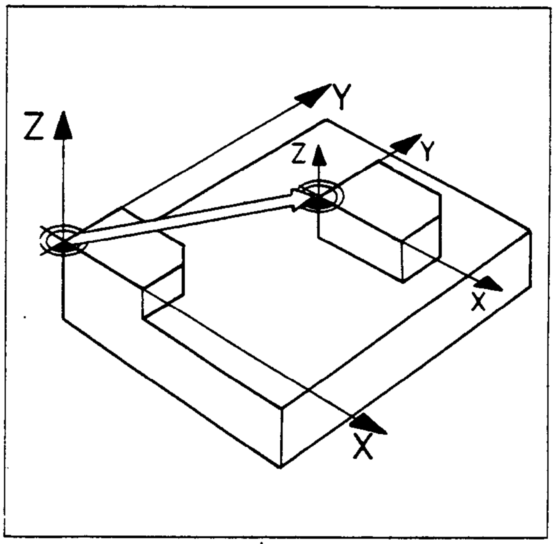
As a result of the tool compensation, the control moves the tool to X+40, Y+50 and Z–5.

Coordinate Transformations
Cycle 7: Datum shift
Datums within the part program

The cycle

You can program a datum shift (also called a zero offset) to any point within a part program. The manually set absolute workpiece datum remains unchanged.

Thus, identical machining steps (e.g. subprograms) can be executed at different positions on the workpiece without having to reenter the program section each time.



Combining with other coordinate transformations

If a datum shift is to be combined with other transformations, the shift has to be made **before** the other transformations!

This enables you to machine a part or detail at different locations with a transformed geometry, (e.g. mirrored, enlarged or reduced, rotated).

Effect

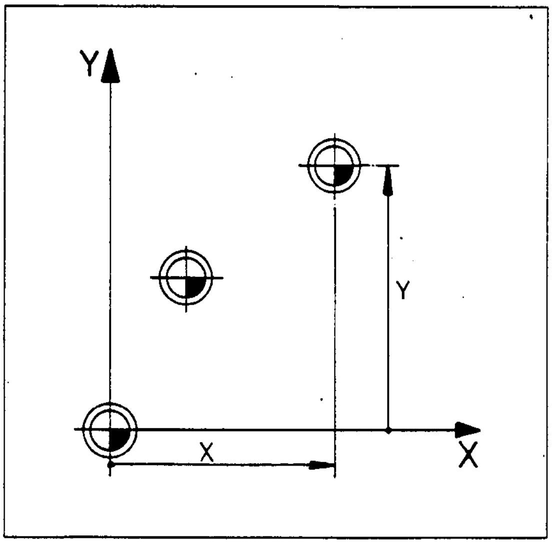
For a datum shift definition, only the coordinates of the new datum are to be entered.

An active datum shift is displayed in the status field. All coordinate inputs then refer to the new datum.

Incremental/ absolute

In the cycle definition the coordinates can be entered as absolute or incremental dimensions:

- **Absolute:** The coordinates of the new datum refer to the manually set workpiece datum. Refer to the center figure.
- **Incremental:** The coordinates of the new datum refer to the last valid datum, which can itself be shifted. Refer to the lower figure.

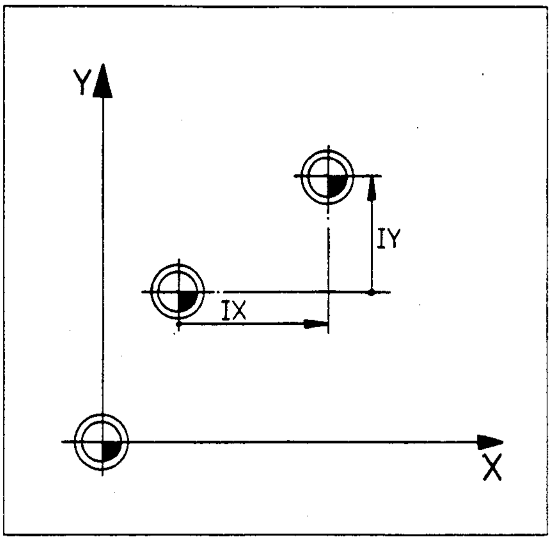


Absolute datum shift

Cancelling the shift

A datum shift is cancelled by entering the datum shift X0/Y0/Z0 ...
Only the "shifted" axes have to be entered.

```
CYCL DEF 7.0 DATUM SHIFT
CYCL DEF 7.1 X+0
CYCL DEF 7.2 Y+0
```



Incremental datum shift

Coordinate Transformations
Cycle 7: Datum shift
Datums within the part program

Selecting the cycle

Initiate the dialog — CYCL DEF ↓ or GOTO 7 ENT

CYCL DEF 7 DATUM SHIFT	ENT Confirm the selected cycle.

Entering the value

SHIFT ?	X Select the axis. Enter the coordinates of the new datum. Y The datum shift is possible in all 4 axes. ENT When shifting in several axes, only confirm entry after entering all the coordinates!

Example

A machining task is to be carried out as a subprogram

a) referred to the set datum X+0/Y+0 and

b) additionally referred to the shifted datum X+40/Y+60.

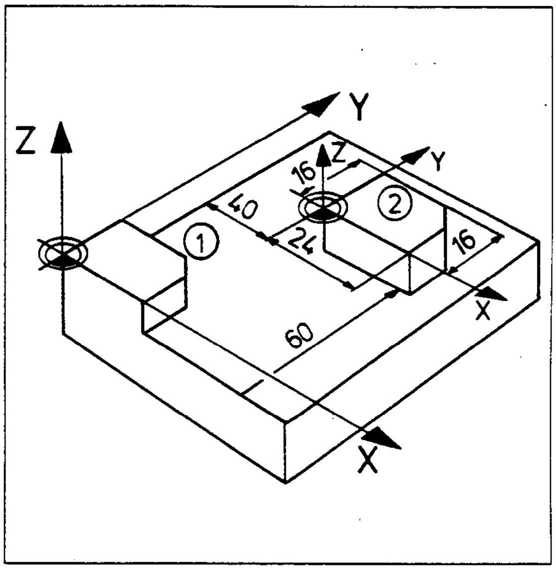
```
CALL LBL 1              Without datum shift ①

CYCL DEF 7.0 DATUM SHIFT
CYCL DEF 7.1 X+40
CYCL DEF 7.2 Y+60

CALL LBL 1              With datum shift ②

CYCL DEF 7.0 DATUM SHIFT
CYCL DEF 7.1 X+0        Datum shift reset
CYCL DEF 7.2 Y+0

L Z+50 FMAX M02
```



Subprogram

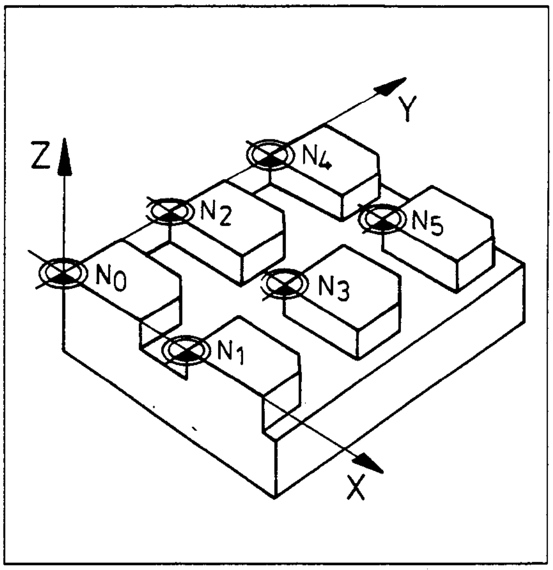
```
LBL 1
L X–10 Y–10 R0 FMAX
L Z+2 FMAX
L Z–5 M36
L X+0 Y+0 RL
L Y+20
L X+25
L X+30 Y+15
L Y+0
L X+0
L X–10 Y–10 R0
L Z+2 FMAX M37
LBL 0
```

Coordinate Transformations
Cycle 7: Datum shift
Datum points from the datum table 0.D

The cycle

In addition to datum shifting within a part program it is also possible to take datum points from a separate datum table.

Datum tables are especially helpful with frequently repeating machining sequences or frequent use of the same datum shift.



Absolute

The datum points from the datum table are effective with their coordinates as absolute values and remain so until a new datum is called.

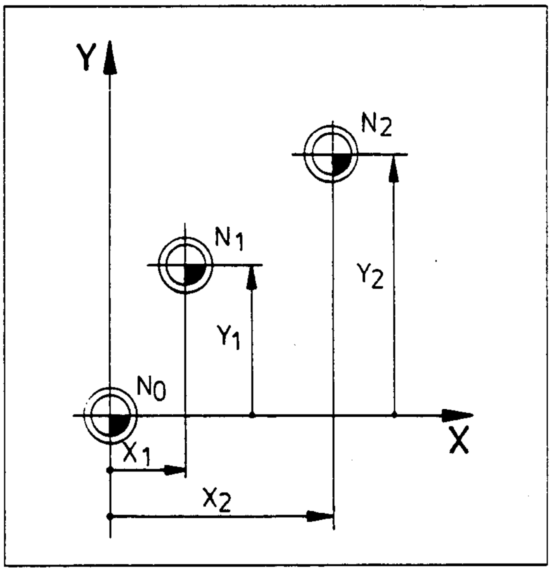
A datum can be shifted in one of two ways, depending on MP 7411.

If MP 7411 = 0:

The coordinate system is shifted by the values in the axes.

If MP 7411 = 1:

The coordinate system is shifted by the values in the axes X, Y and Z. However, a datum shift in a rotary axis also rotates the coordinate system of the corresponding working plane. The datum shift always precedes the rotation.



Cancelling the shift

The datum shift can be cancelled either
- in the conventional way
- in the datum table
  (see following page "Datum table").

To activate the datum table

Enter the datum table as program number 0 in the program directory (PGM NR).

To insert lines

Use the CYCL DEF key to insert additional lines. Each line is automatically given a sequential number.

To erase lines

Use the cursor keys to select the unwanted table and press DEL □.

Editing function

To jump to a certain datum, press GOTO □ and enter the datum number.
Use GOTO □ and the cursor keys to jump to the beginning or end of tables or columns.

CYCL DEF

Coordinate Transformations
Cycle 7: Datum shift
Datum points from the datum table 0.D

Cycle selection

Initiate dialog — CYCL DEF ↓ or GOTO □ 7 ENT

CYCL DEF 7 DATUM SHIFT	ENT Select cycle.

Input

SHIFT ?	□ Enter datum number D. ENT Confirm entry.

Example

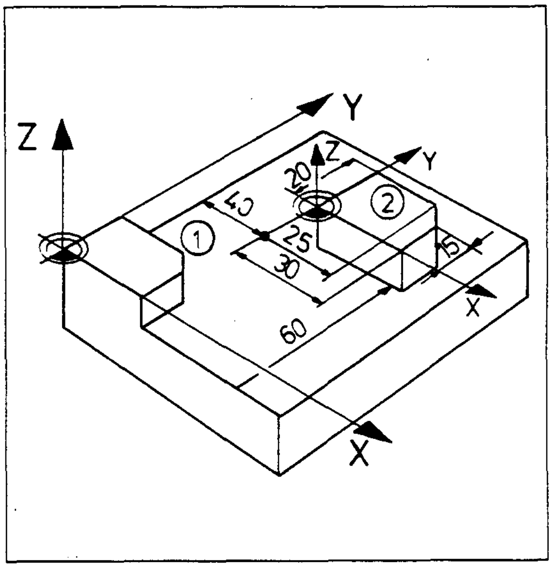
A machining sequence in the form of a subprogram is to be
a) referenced to the datum X+0/Y+0 and
b) also referenced to the shifted datum X+40/Y+60.

Datum table

D	X	Y	Z	C
0	+ 0	+ 0	+0	+0
1	+40	+60	+0	+0
[END]				

Program

CALL LBL 1	Without datum shift ①
CYCL DEF 7.0 DATUM SHIFT CYCL DEF 7.1 # 1	
CALL LBL 1	With datum shift ②
CYCL DEF 7.0 DATUM SHIFT CYCL DEF 7.1 # 0	Cancel datum shift
L Z+50 M02	



Subprogram

```
LBL 1
L X–20 Y–20 R0 FMAX
L Z+2
L Z–5 M36
L X+0 Y+0 RL
L Y+20
L X+25
L X+30 Y+15
L Y+0
L X+0
L X–20 Y–20 R0
L Z+2 FMAX M37
LBL 0
```

Coordinate Transformations
Transferring numerical values between the NC program and the datum table 0.D

M functions M38/M39

In addition to the datum shift function with values from the datum table 0.D, it is also possible to use the M function M38 to transfer the values located in a datum table to Q parameters in an NC program.

The M function M39 transfers Q parameters from an NC program to the datum table 0.D.

Both M functions become effective at the beginning of the NC block.

Q parameters Q80 to Q84

The parameters Q80 to Q84 receive the following information:

Q80 = Datum number in the datum table 0.D
Q81 = Coordinate value in the X axis
Q82 = Coordinate value in the Y axis
Q83 = Coordinate value in the Z axis
Q84 = Coordinate value in the IV axis

Example with M38

You wish to move the axes X, Y and Z in a straight line to the coordinate values filed under the datum number D = 4.

FN0: Q80 = +4	Define the datum number.
L X+Q81 Y+Q82 Z+Q83 R0 M38	

Example with M39

You wish to transfer the uncompensated measured values for the X, Y, Z and IV axes from the programmable probe function "Reference plane" to the datum table 0.D under the datum number D = 5 (see also the section "Programmed Probing").

TCH PROBE 0.0 REF. PLANE Q10 Z- TCH PROBE 0.1 X+0 Y+0 Z+10	Probe with the Z axis in negative direction and move to the starting position.
FN0: Q80 = +5	Define the datum number.
FN0: Q81 = Q115 FN0: Q82 = Q116 FN0: Q83 = Q117 FN0: Q84 = Q118	Copy the uncompensated measured values for X, Y, Z and IV to the transfer parameters Q81 to Q84.
M39	Transfer the uncompensated measured values of the probe to the datum table 0.D.

Error messages

If under parameter Q80 a datum number that does not exist in the datum table is addressed with the M function M38 or M39, this is answered with the error message:

DATUM NOT DEFINED

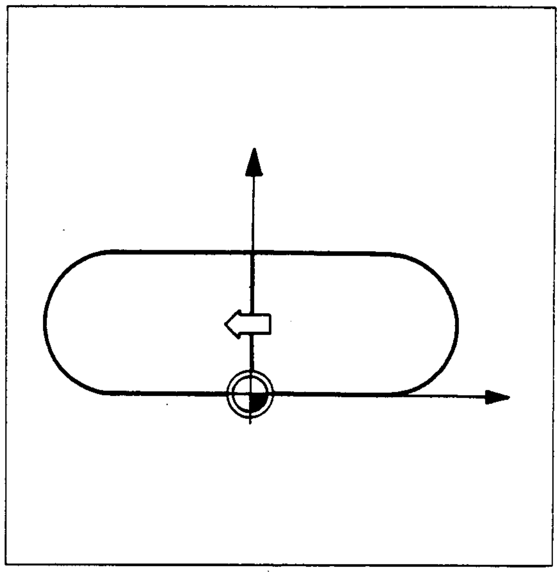
If you try to use M function M39 to transfer values to the datum table 0.D and the table is protected against editing and erasure (see the section "File Management"), this is answered with the error message:

PROTECTED PGM!

Coordinate Transformations
Cycle 8: Mirror image

The cycle

The direction of an axis is reversed when it is mirrored. The sign is reversed for all coordinates of this axis. The result is a mirror image of a progammed contour or of a hole pattern.
Mirroring is only possible in the machining plane. You can mirror in one axis or both axes simultaneously.



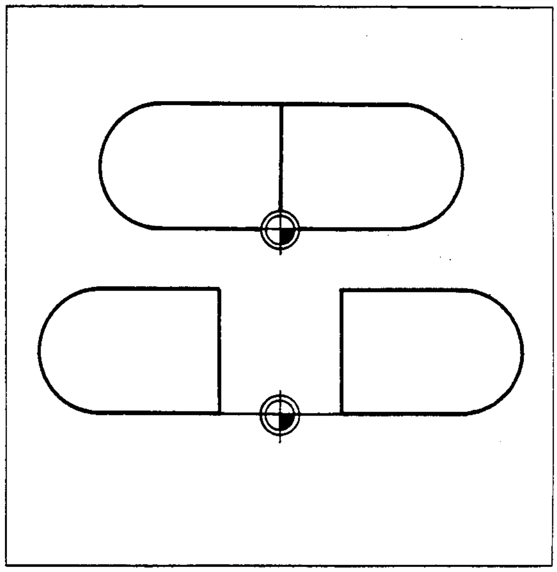
Activation

The mirror image is immediately valid upon definition. The mirrored axes can be recognized by the highlighted axis designations in the status display for the datum shift.

Mirroring is performed at the current datum. The datum must therefore be shifted to the required position before a "mirror image" cycle definition.

Datum position

1. If the datum is on the part contour, the part is only mirrored across the axis.
2. If the datum is outside the contour, the part is also moved!



Cancelling the mirror image

The mirror image cycle is cancelled by entering the mirror image cycle and responding to the dialog query "mirror image axis" with "NO ENT":

CYCL DEF 8.0 MIRROR IMAGE
CYCL DEF 8.1

Mirrored axes

Enter the axis or axes to be mirrored.
The tool axis cannot be mirrored.

Rotating direction

The rotating direction of the C axis remains the same when one or more axes are mirrored.

Coordinate Transformations
Cycle 8: Mirror image

Selecting the cycle

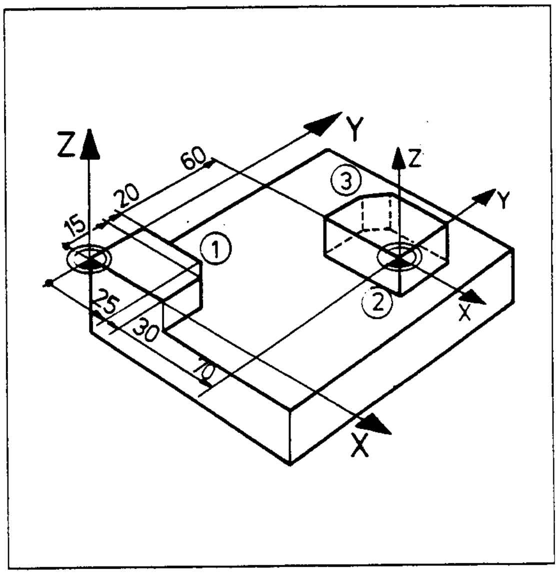
Initiate the dialog — CYCL DEF ↓ or GOTO □ 8 ENT

CYCL DEF 8 MIRROR IMAGE	ENT Confirm the selected cycle.

MIRROR IMAGE AXIS ?	X Enter the axis to be mirrored, e. g. X. Y Enter the second axis to be mirrored if applicable, e. g. Y. END □ Confirm the axes and always terminate the input with "END □".

Example

A program section (subprogram 1) is to be executed once – at position X+0/Y+0, and also mirrored once in X at position X+70/Y+60.




```
TOOL DEF 1 L+0 R5
TOOL CALL 1 Z U+0.5

CALL LBL 1 REP                   Not mirrored ①

                                 Mirrored
                                 execution:
                                 sequence

CYCL DEF 7.0 DATUM SHIFT         Datum shift ②
CYCL DEF 7.1 X+70
CYCL DEF 7.2 Y+60

CYCL DEF 8.0 MIRROR IMAGE        Mirror image ③
CYCL DEF 8.1 X
CALL LBL 1                       Subprogram call

CYCL DEF 8.0 MIRROR IMAGE        Reset mirror image
CYCL DEF 8.1
CYCL DEF 7.0 DATUM SHIFT         Cancel datum shift
CYCL DEF 7.1 X+0
CYCL DEF 7.2 Y+0

L Z+50 FMAX M02                  Retract, return jump
```

Subprogram:

```
LBL 1
L X–10 Y–10 R0 FMAX
L Z+2 FMAX
L Z–5 M36
L X+0 Y+0 RL
L Y+20
L X+25
L X+30 Y+15
L Y+0
L X+0
L X–10 Y–10 R0
L Z+2 FMAX M37
LBL 0
```

Note

For correct machining according to the drawing, it is absolutely necessary that the sequence of cycles shown in the above execution be retained!

Coordinate Transformations
Cycle 10: Coordinate system rotation

The cycle

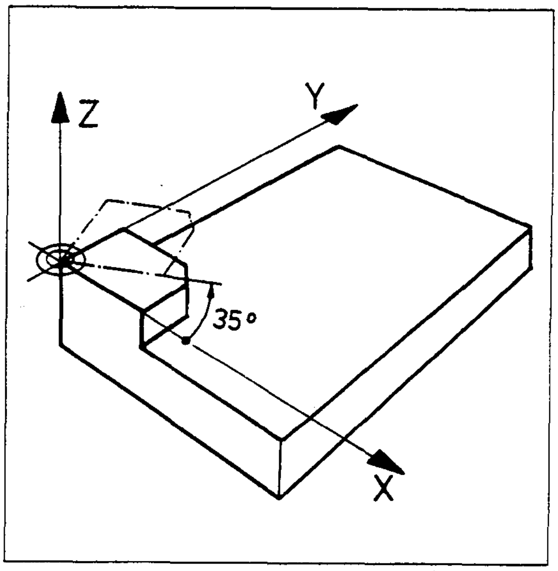
The coordinate system can be rotated in the machining plane about the current datum in a program.

Activation

Rotation is effective without being called and is also active in the operating mode "Positioning with MDI".

Rotation

To rotate the coordinate system, you only have to enter the rotation angle ROT.

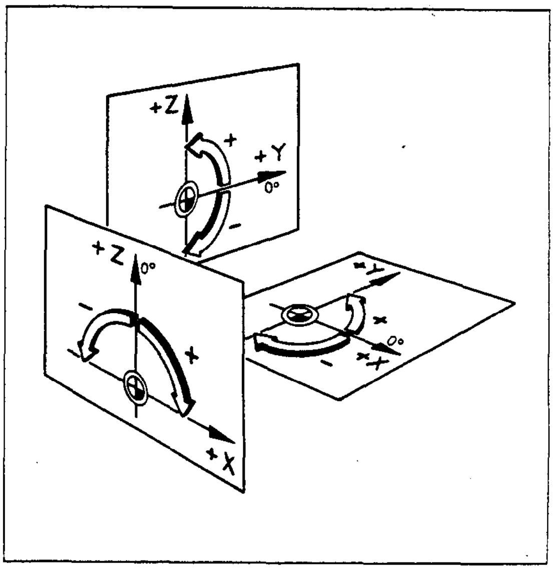


Planes

XY plane: +X axis = 0° (standard)
YZ plane: +Y axis = 0°
ZX plane: +Z axis = 0°

All coordinate inputs following the rotation are then referenced to the rotated coordinate system.

The rotation angle is entered in degrees (°).
Input range: –360° to +360° (absolute or incremental).



Activating the rotation

CYCL DEF 10.0 ROTATION
CYCL DEF 10.1 ROT+35

The active rotation angle is indicated by "ROT" in the status display.

Cancelling the rotation

A rotation is cancelled by entering the rotation angle 0°.

CYCL DEF 10.0 ROTATION
CYCL DEF 10.1 ROT+0

Coordinate Transformations
Cycle 10: Coordinate system rotation

Selecting the cycle

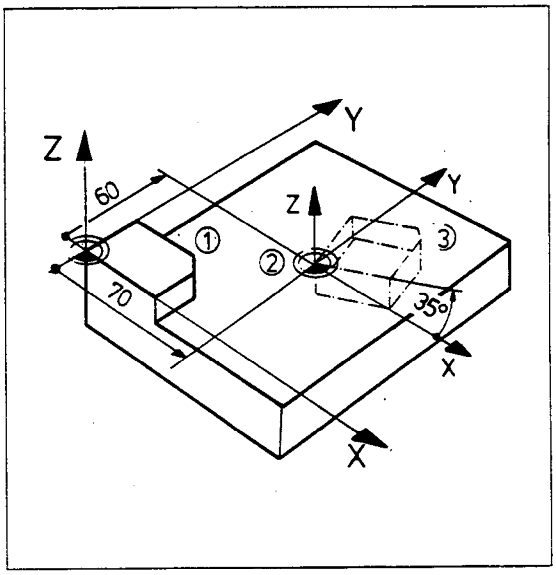
Initiate the dialog — CYCL DEF ↑ or GOTO □ 1 0 ENT

CYCL DEF 10 ROTATION	ENT Confirm the selected cycle.

ROTATION ANGLE ?	Enter the rotation angle. I Incremental/absolute? ENT Confirm entry.

Example

A program section (subprogram 1) is to be executed:
once based on datum X+0/Y+0,
a second time based on datum X+70 Y+60.



TOOL DEF 1 L0 R5 TOOL CALL 1 Z U+0.5	
CALL LBL 1	Non-rotated execution ①
CYCL DEF 7.0 DATUM SHIFT CYCL DEF 7.1 X+70 CYCL DEF 7.2 Y+60	Rotated execution. Sequence: 1. Datum shift ②
CYCL DEF 10.0 ROTATION CYCL DEF 10.1 ROT+35	2. Rotation ③
CALL LBL 1	3. Subprogram call
CYCL DEF 10.0 ROTATION CYCL DEF 10.1 ROT 0	Reset rotation
CYCL DEF 7.0 DATUM SHIFT CYCL DEF 7.1 X+0 CYCL DEF 7.2 Y+0	Cancel datum shift
L Z+200 FMAX M02	Return jump to first block of the main program

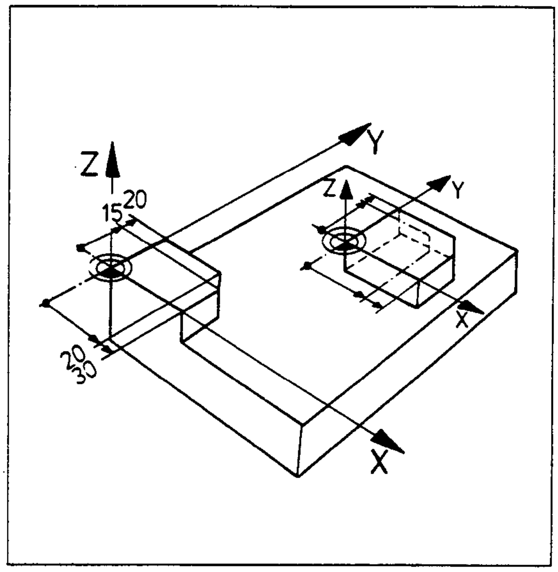
Subprogram

The associated subprogram (see cycle 7, Datum shift) is programmed after M02.

Coordinate Transformations
Cycle 11: Scaling

The cycle

Contours can be enlarged or reduced with this cycle. This permits generation of contours geometrically similar to an original without reprogramming, and also use of shrinkage and growth allowances.



Scaling is effective – depending on the specified machine parameters – either in the machining plane or in the three main axes (see index General Information, MOD Functions, User parameters).

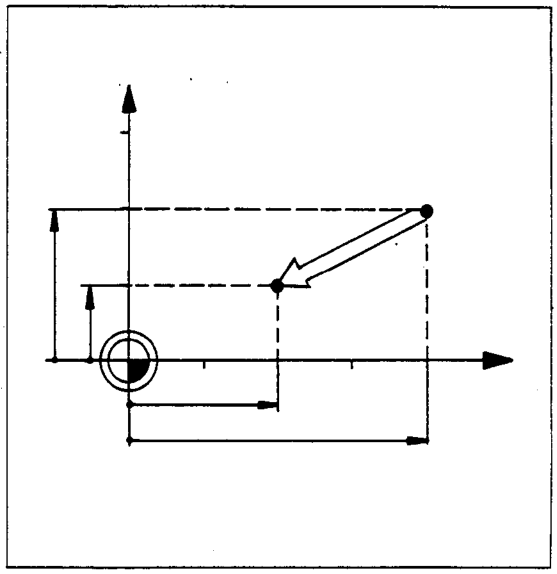
Activation

Scaling is effective immediately, without being called. Scaling factors greater than 1 result in magnification, factors between 0 and 1 result in reduction.

SCL factor

The scaling factor SCL (scaling) is entered to magnify or reduce a contour.
The control applies this factor to all coordinates and radii either in the machining plane or (depending on MP 7410; see index General Information, MOD Functions, User parameters) in all three axes X, Y and Z. The factor also affects dimensions in cycles.

Input range: 0.000 001 to 99.999 999.



Datum position

It is helpful to locate the datum on an edge of the subcontour. This way, the datum of the coordinate system is retained during a reduction or magnification as long as it is not subsequently moved or if the move is programmed before the scaling factor.

Activating scaling

CYCL DEF 11.0 SCALING
CYCL DEF 11.1 SCL 0.8

Cancelling scaling

The scaling cycle is cancelled by redefining it with the factor 1:

CYCL DEF 11.0 SCALING
CYCL DEF 11.1 SCL 1.0

Coordinate Transformations
Cycle 11: Scaling

Selecting the cycle

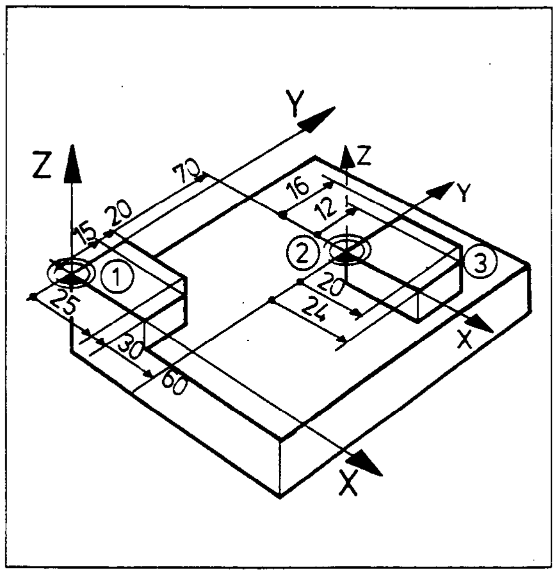
Initiate the dialog — CYCL DEF ↑ or GOTO □ 1 1 ENT

CYCL DEF 11.0 SCALING	ENT Confirm the selected cycle.

FACTOR ?	□ Enter the scaling factor. ENT Confirm entry.

Example

A program section (subprogram 1) is to be executed one time based on the manually set datum X+0/Y+0, and one time based on X+60/Y+70 with the scaling factor 0.8.



TOOL DEF 1 L+0 R5
TOOL CALL 1 Z U+0.5

CALL LBL 1	Execution in original size ①
	Execution with scaling factor. Sequence:
CYCL DEF 7.0 DATUM CYCL DEF 7.1 X+60 CYCL DEF 7.2 Y+70	1. Shift datum ②
CYCL DEF 11.0 SCALING CYCL DEF 11.1 SCL 1.0	2. Define scaling factor ③
CALL LBL 1	3. Call subprogram (scaling factor effective)
CYCL DEF 11.0 SCALING CYCL DEF 11.1 SCL 1.0 CYCL DEF 7.0 DATUM CYCL DEF 7.1 X+0 CYCL DEF 7.2 Y+0	Cancel transformations
L Z+200 FMAX M02	Retract, return jump

Subprogram

The corresponding subprogram (see cycle 7, Datum shift) is programmed after M02.

Other Cycles
Cycle 9: Dwell time

The cycle

In a program which is being run, the next block will be executed only after the end of the programmed dwell time. Modal conditions, such as radius compensation, are not affected.

Activation

The dwell cycle is valid immediately upon definition, without being called.

CYCL DEF 9.0 DWELL TIME
CYCL DEF 9.1 DWELL 0.500

Effect

Dwell time can be used to delay the start or end of any erosion process.

Input range

The dwell time is specified in seconds. Input range: 0 to 30 000 s (≙ 8.3 hours)

Cycle definition

Initiate the dialog CYCL DEF ↑ or GOTO □ 9 ENT

CYCL DEF 9.0 DWELL TIME	ENT Confirm the selected cycle.

DWELL TIME IN SECS. ?	□ Enter desired dwell time, in seconds. ENT Confirm entry.

The cycles

Machining procedures that you have programmed – such as special eroding cycles, curve eroding, or geometry modules – can be created as callable main programs and used like fixed cycles. They can be called from any program with a cycle call. They can thus help speed up programming and improve safety, since you are using proven modules.

Cycle 12 PGM CALL

A callable program defined as a cycle becomes in essence a fixed cycle.

It can be called with

CYCL CALL (separate block) or

M99 (blockwise) or

M89 (modally).

Initiate the dialog — CYCL DEF ↑ or GOTO □ 1 2 ENT

Entering the cycle selection

CYCL DEF 12 PGM CALL	ENT	Confirm the selected cycle.

PROGRAM NUMBER ?	□ ENT	Program number

Example

The callable program 50 is to be called from program 5.

Program:
BEGIN PGM 5 MM
.
.

CYCL DEF — CYCL DEF 12.0 PGM CALL
CYCL DEF 12.1 PGM 50 — Definition: "Program 50 is a cycle"

CYCL CALL — L X+20 Y+50 FMAX M99 — Call program 50
.
.

END PGM 5 MM

Parametric Programming
Overview

Parametric programming

Parametric programming expands the capabilities of the control enormously and offers features such as:

- Variable eroding programs
- Processing of mathematical curves (e.g.: sine wave, ellipse, parabola, hyperbola)
- Programs for machining families of parts
- 3D programming for mold making

Basic functions

The mathematical and logical functions listed at the right are available for programming.

FN 0:	ASSIGN
FN 1:	ADDITION
FN 2:	SUBTRACTION
FN 3:	MULTIPLICATION
FN 4:	DIVISION
FN 5:	SQUARE ROOT
FN 6:	SINE
FN 7:	COSINE
FN 8:	ROOT-SUM OF SQUARES
FN 9:	IF EQUAL, JUMP
FN 10:	IF UNEQUAL, JUMP
FN 11:	IF GREATER, JUMP
FN 12:	IF LESS, JUMP
FN 13:	ANGLE
FN 14:	ERROR CODE
FN 15:	PRINT
FN 16:	INDEXED DATA ASSIGNMENT

Computation time

The time required for one computing step – depending on the workload on the processor – can reach the millisecond range.

For this reason, very many computations and very small displacements may cause the machine axes to be halted. In this case you have to make a compromise between high surface definition (many computations, small displacements) and efficient machining.

Variable addresses with parameters

The program data shown at the right can be kept variable by using the Q parameters:
Enter a Q parameter instead of a specific number.

Example for variable positioning:
instead of L X+20.25 you write L X+Q21.

The parameter value for Q21 must be computed in the program or be defined before it is called.

Nominal positions	L X+Q21 Y+Q22
Circle data	CC X+Q1 Y+Q2 C X+Q10 Y+Q20 CT X+Q11 Y+Q21 RND Q1 CR X+Q21 Y+Q22 R Q62
Feed rate	F Q10
Tool data	TOOL DEF 1 L+Q1 R Q2 TOOL CALL Q5 Z U Q6
Conditional jump	IF+Q10 GT+0 GOTO LBL Q30
Cycle data	CYCL DEF 17 DISK AXIS Z+Q1 M36 EXPANSION RADIUS Q2 EXPANSION MODE 0

Inch dimensions

Programs using parameters as jump address (e.g. GOTO LBL Q10) are not to be switched from mm to inches or vice versa, because the contents of the Q parameters are also converted during switchover, which would result in false jump addresses.

Parametric Programming
Selection

Selecting basic functions

Q

After pressing "Q", the functions can be selected either with the vertical cursor keys or with "GOTO □", the associated function number and "ENT".

Defining parameters

A parameter is designated by the letter Q and any number between 0 and 99.

Specific numerical values can be allocated to the parameter either directly or with mathematical and logical functions. Parameter contents can also have a negative sign. Positive signs need not be programmed.

Starting values

Parameters must be defined before they can be used. When program run is started, all parameters are automatically assigned the value 0 if machine parameter MP 7300 = 0. If MP 7300 = 1 and power is interrupted, the parameters Q0 to Q99 are stored and can be erased only after selecting the program by pressing PGM NR, entering the program number, and confirming with ENT.

Examples of defined parameters:
Q1 = +1.5
Q5 = +Q1
Q9 = +Q1 * +Q5

Notation

The notation corresponds to the standard computer format:
The operands and the operator are on the right, the desired result on the left. Consider the entire line as a mathematical operation and not as an equation!

Here also use the "ENT" key to continue the dialog within one program line.

Example

The following multiplication is to be entered:

$Q10 = Q5 \cdot 1.7$

Initiate the dialog — Q

FN 0: = ASSIGN	Q ↓ or GO TO 3 ENT	Select multiplication (operation).

FN 3: MULTIPLICATION	ENT	Entry into the function.

PARAMETER NUMBER FOR RESULT?	10 ENT	Parameter for result.
FIRST VALUE OR PARAMETER?	Q 5 ENT	1st operand (parameter).
SECOND VALUE OR PARAMETER?	1.7 ENT	2nd operand.

FN 3: Q10 = +Q5 * +1.7	Finished program line.

Q10 is assigned the result when the operation is executed; the contents of Q5 are retained.

Parametric Programming
Algebraic functions

FN 0: Assignment

This function assigns a parameter either a numerical value or another parameter.

The assignment corresponds to an equal sign.

Example:
FN 0: Q5 = +65.432

Q5 = +Q12
Q5 = −Q13

FN 1: Addition

This function defines a certain parameter to be the sum of two parameters, two numbers or one parameter and one number.

FN 1: Q17 = +Q2 + +5

Q17 = +5 + +7
Q17 = +5 + −Q12
Q17 = −Q4 + +Q8
Q17 = +Q17 + +Q17

FN 2: Subtraction

This function defines a certain parameter to be the difference between two parameters, two numbers or one parameter and one number.

FN 2: Q11 = +5 − +Q34

Q11 = +5 − +7
Q11 = +5 − −Q12
Q11 = +Q4 − +Q8
Q11 = +Q11 − −Q11

FN 3: Multiplication

This function defines a certain parameter to be the product of two parameters, two numbers or one parameter and one number.

FN 3: Q21 = +Q1 * +60

Q21 = +5 * +7
Q21 = +5 * −Q12
Q21 = +Q4 * −Q8
Q21 = +Q21 * +Q21

FN 4: Division

This function defines a certain parameter to be the quotient of two parameters, two numbers or one parameter and one number.

Division by 0 is not possible!

FN 4: Q12 = +Q2 DIV +62

Q17 = +5 DIV +7
Q17 = +5 + DIV −Q12
Q17 = +Q4 DIV +Q8

FN 5: Square root

This function defines a certain parameter to be the square root of one parameter or one number. The operand must be positive.

FN 5: Q98 = SQRT +2

Q98 = SQRT +Q12
Q98 = SQRT −Q70

Sign for operands

Parameters with negative signs can be used in equations.

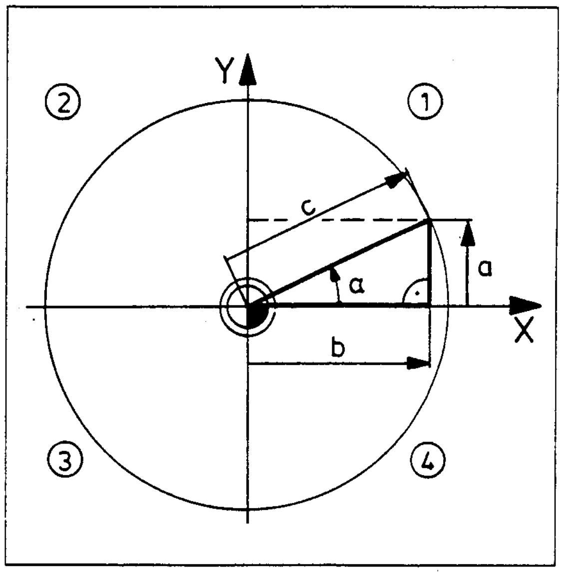
Q11 = +5 − −Q34

E. G. subtraction can be obtained from an addition and vice versa. This also applies for other operations.

Parametric Programming
Trigonometric functions

Basics of trigonometry

A circle with radius c is divided symmetrically into four quadrants ① to ④ by the two axes X and Y. If the radius c forms the angle α with the X-axis, the two components a and b of the right-angled triangle depend upon angle α



Defining the trigonometric functions

$\sin\alpha = \dfrac{\text{opposite side}}{\text{hypotenuse}} = \dfrac{a}{c}$ or $a = c \cdot \sin\alpha$

$\cos\alpha = \dfrac{\text{adjacent side}}{\text{hypotenuse}} = \dfrac{b}{c}$ or $b = c \cdot \cos\alpha$

$\tan\alpha = \dfrac{\sin\alpha}{\cos\alpha} = \dfrac{\text{opposite side}}{\text{adjacent side}} = \dfrac{a}{b}$

Length of one side

According to the Pythagorean theorem:
$c^2 = a^2 + b^2$ or $c = \sqrt{a^2 + b^2}$

Table for preceding sign and angle range

Function	Quadrant ①	②	③	④
sin α	+	+	−	−
cos α	+	−	−	+
tan α	+	−	+	−
Angle	0° – 90°	90° – 180°	180° – 270°	270° – 360°

FN 6: Sine

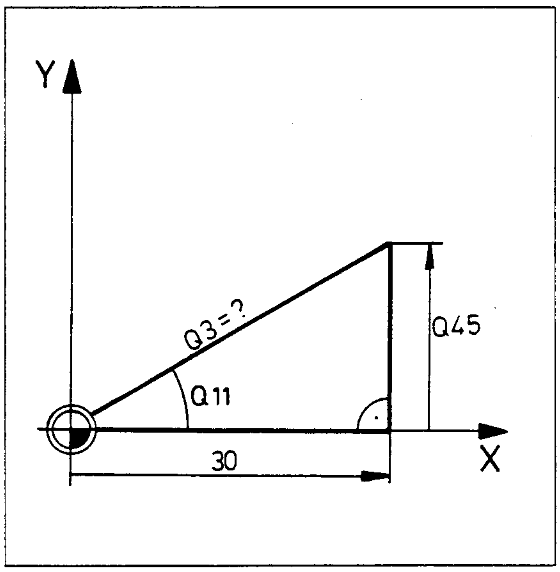
A parameter is defined as the **sine** of an angle, whereby the angle can be a number or a parameter (unit of measurement of the angle: degrees).
Q44 = sin Q11

FN 6: Q44 = SIN + Q11

FN 7: Cosine

A parameter is defined as the **cosine** of an angle, whereby the angle can be a number or a parameter (unit of measurement of the angle: degrees).
Q81 = cos Q11

FN 7: Q81 = cos + Q11



FN 8: Root sum of squares

A parameter is computed as the **square root** of the sum of squares of two numbers or parameters (LEN = length).
$Q3 = \sqrt{Q45^2 + 30^2}$

FN 8: Q3 = +Q45 LEN+30

Parametric Programming
Trigonometric functions

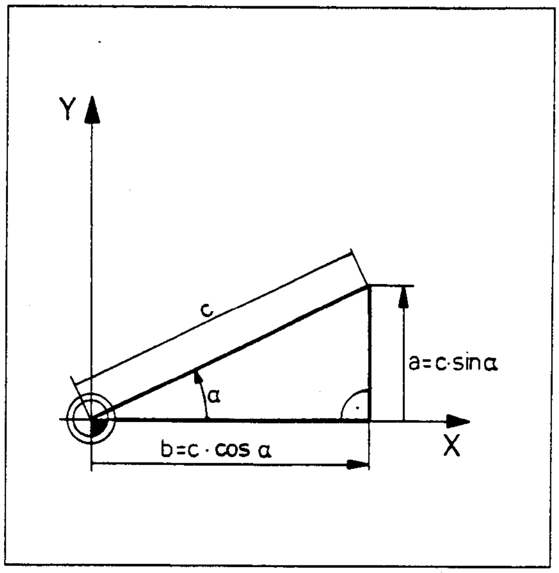
Angles from line segments or trigonometric functions

According to the definitions of the angular functions, either the angular functions sin α and cos α or the lengths of sides a and b can be used to determine tan α:

$$\tan\alpha = \frac{\sin\alpha}{\cos\alpha} = \frac{a}{b}\ .$$

The angle α is therefore

$$\alpha = \arctan\frac{\sin\alpha}{\cos\alpha} = \arctan\frac{a}{b}\ .$$



Unambiguous angle

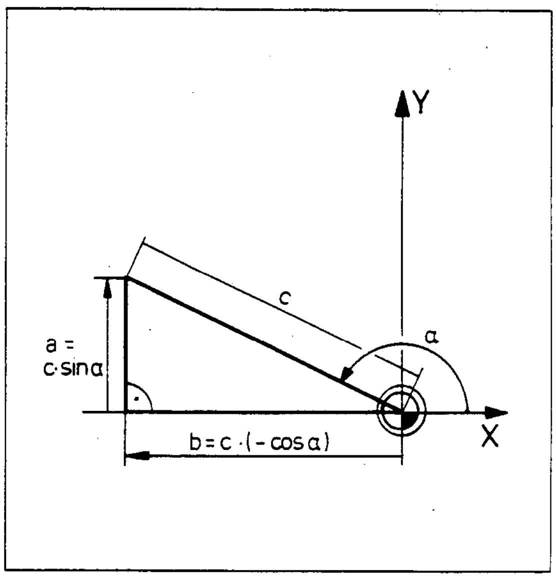
If the value of sin α or the side a is known, two possible angles always result:

Example: $\sin\alpha = 0.5$
$\alpha_1 = +30°$ and $\alpha_2 = +150°$.

To determine angle α unambiguously, the value for cos α or side b is required. If this value is known, an **unambiguous** angle α is the result:

Example: $\sin\alpha = 0.5$ and $\cos\alpha = 0.866$
$\alpha = +30°$,

$\sin\alpha = 0.5$ and $\cos\alpha = -0.866$
$\alpha = +150°$.



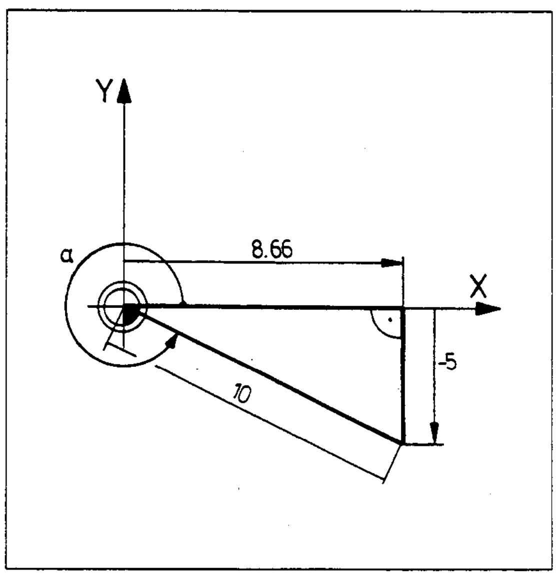
FN 13: Angle

This function assigns a parameter the **angle** from a sine and cosine function, or from the two legs of the right-angled triangle.

$$\tan\alpha = \frac{\sin\alpha}{\cos\alpha} = \frac{a}{b} = \frac{-5}{8.66}$$

$$\alpha = \arctan\left(\frac{-5}{8.66}\right)$$

FN 13: Q11 = –5 ANG +8.66



Parametric Programming
Conditional/unconditional jumps

IF:
If-then jump

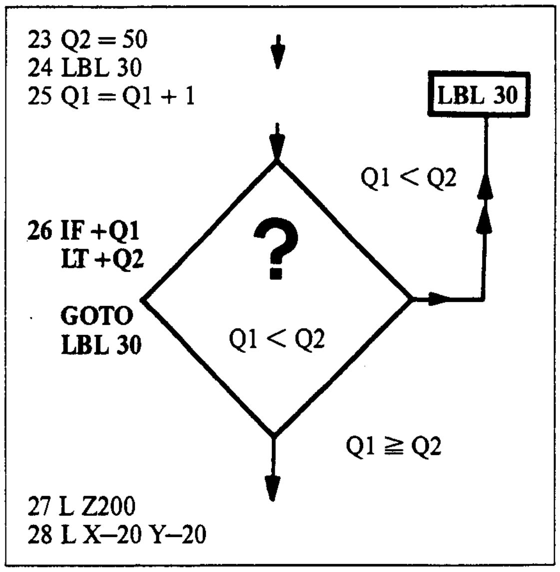
With the parameter functions FN 9 to FN 12, you can compare one parameter with another parameter or with a given number (e.g. a maximum value).

Depending on the result of this comparison, a jump to a certain label in the program can be programmed (conditional jump):

If the programmed IF condition is fulfilled, a jump is performed; if the condition is not fulfilled, the next block (following IF . . .) will be executed.

Program call

If you write a program call behind the called program label, a jump can be made to another program.
(Program calls are for example PGM CALL or cycle 12).



	Examples:	Decision criteria:
Equation FN 9: =	FN 9: IF + Q1 EQU + 360 GOTO LBL 30	A parameter is equal to a value or a second parameter, e.g. Q1 = Q2 or in the example: Q1 has the value 360.000.
Inequalities FN 10: ≠	FN 10: IF + Q1 NE + Q2 GOTO LBL 2	A parameter is not equal to a value or a second parameter, e.g. Q1 ≠ Q2
FN 11: >	FN 11: IF + Q1 GT + 360 GOTO LBL 17	A parameter is greater than a value or a second parameter, e.g. Q1 > Q2. Also possible: greater than zero, i.e. positive.
FN 12: <	FN 12: IF + Q1 LT + Q2 GOTO LBL 3	A parameter is less than a value or a second parameter, e.g. Q1 < Q2. Also possible: less than zero, i.e. negative.

Unconditional jumps

You can also program **unconditional jumps** to a label with the parameter functions FN 9 to FN 12.

Example:	Decision criterion:
FN 9: IF 0 EQU 0 GOTO LBL 30	The condition is **always** fulfilled, i.e. an **unconditional** jump is performed.

Abbreviations

EQU: equal to
NE: not equal to
GT: greater than
LT: less than

FN 14:
Error code

You can call error messages and dialog texts of the machine manufacturer from the PLC EPROM with FN 14. To call, enter the error code number between 0 and 499.
The error message terminates program run.
The program must be restarted after the error has been corrected.

The messages are allocated as follows:

Error number	Screen display
0 ... 299	ERROR 0 ... ERROR 299
300 ... 399	PLC ERROR 01 ... PLC ERROR 99 (or dialog determined by the machine tool manufacturer).
400 ... 483	DIALOG 1 ... 83 (or dialog determined by the machine tool manufacturer).
484 ... 499	USER PARAMETER 15 ... 0 (or dialog determined by the machine tool manufacturer).

Example:

FN 14: ERROR = 100

FN 15:
Print

This function outputs current Q-parameter values through the RS-232-C serial interface. You can also enter numerical values between 0 and 499 instead of Q parameters. These values call error messages and dialog texts which are stored in the PLC EPROM and are allocated as with FN 14. You can enter combinations of up to six Q parameters and numerical values.

Example:

FN 15: PRINT Q1/20/Q9/0/Q17/Q330

FN 16:
Indexed data assignment

With indexed data assignment you can access the elements of a list (e.g. "two-times gap" values) through a base parameter with the aid of a pointer variable.

Example:

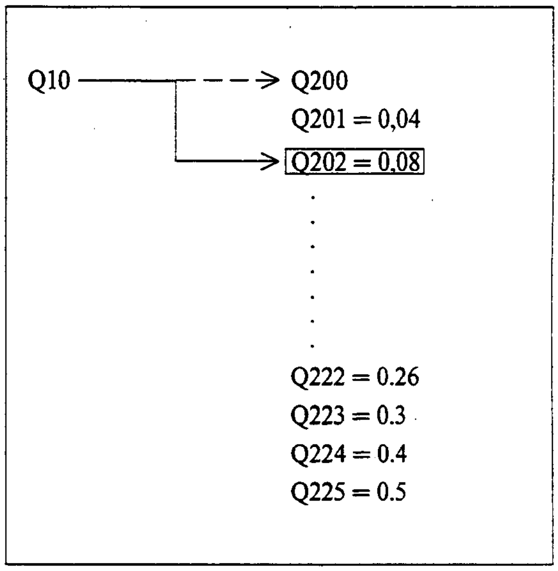
Q99 = 2

pointer parameter

Q10 = Q200 (Q99)

base parameter

→ Q10 = < Q203 > = 0.08



Q0 to Q89 Vacant Q parameters

You can use parameters Q80 to Q89 in the NC program for calculations, i.e. they can be both read and overwritten.

Q80 to Q84 Datum table 0.D

Parameters Q80 to Q84 are used together with the M functions M38 (reading values from the datum table 0.D) and M39 (writing values to the datum table 0.D) to transfer data between the NC program and the datum table.

Function	Parameter
Number of the datum in the datum table (0 to 99)	Q80
X-axis coordinate	Q81
Y-axis coordinate	Q82
Z-axis coordinate	Q83
C-axis coordinate	Q84

No erosion tables Q90 to Q99

If the control has no erosion tables, the erosion-specific Q parameters (Q90 to Q99) for generator settings and other erosion functions are determined by the machine tool builder.

Erosion tables active Q96 to Q255

If the control has erosion tables, the parameters Q96 to Q255 have the following meanings:

Function	Parameter
Additional erosion parameters (function determined by the machine tool builder)	Q96 to Q98
Momentary power stage LS	Q99
Highest power stage LS	Q150
Lowest power stage LS	Q151
Active erosion parameter table TAB	Q152
Minimum undersize UNS [mm] of the lowest power stage	Q154
Two-times gap 2G [mm] of the lowest power stage	Q155
Two-times gap 2G [mm] of the highest power stage	Q156
Two-times gap 2G [mm] from the lowest to the highest power stage	Q201 to Q225
Minimum undersize UNS [mm] from the lowest to the highest power stage	Q251 to Q255

Q100 to Q107

Q100 to Q107 can be used to transfer Q parameter values from the integrated PLC into an NC program.

Eroding with time limit

If the control jumps from a subprogram or an OEM cycle back into the main program, then Q153 = 0 is set.

If the erosion time limit expires, the disk cycle is interrupted and Q153 = 1 is set.

When cycle 2 is completed, Q153 = 2 is set.

Parametric Programming
Q Parameters with special functions

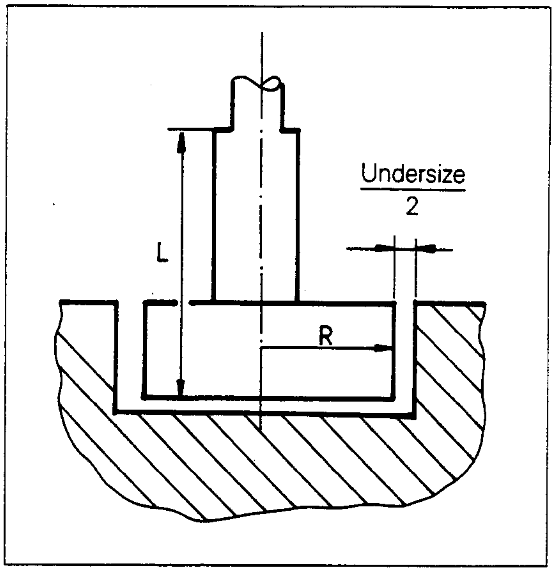
Following electrode Q157

If a following electrode is given during TOOL CALL, Q157 is assigned the value 1. Otherwise it receives the value from user parameter MP 2040.

Tool data Q108, Q158, Q159, Q160

The control files the tool data of the tool which was last called under certain Q parameters.

Q108: Electrode radius from TOOL DEF
Q158: Electrode undersize from TOOL CALL
Q159: Length of the electrode from TOOL DEF
Q160: Electrode number from TOOL CALL



Q109 Tool axis

The control stores the current tool axis (from TOOL CALL) in parameter Q109:
Different machines alternately use the X,Y or Z axis as the tool axis.
On these machines it is helpful when the current tool axis can be requested in the machining program; this makes program branching in user cycles possible.

Current tool axis	Parameter
no tool axis called	Q109 = −1
X axis is called	Q109 = 0
Y axis is called	Q109 = 1
Z axis is called	Q109 = 2
C axis is called	Q109 = 3

Q110 C-axis rotation on/off

The value in parameter Q110 specifies the last M function issued for the direction of non-controlled rotation of the C axis:

M function	Parameter
no M spindle function	Q110 = −1
M03	Q110 = 0
M04	Q110 = 1
M05, if M03 was previously issued	Q110 = 2
M05, if M04 was previously issued	Q110 = 3

Q111
Flushing on/off

Parameter Q111 indicates whether the flushing was switched on or off.

Meaning:	Parameter
M08 flushing switched on	Q111 = 1
M09 flushing switched off	Q111 = 0

Q113
mm/inch dimensions

Parameter Q113 indicates whether the NC program at the highest program level (in cases of sub-programming with PGM CALL) contains mm or inch dimensions.

Meaning:	Parameter
mm dimensions	Q113 = 0
inch dimensions	Q113 = 1

Parameters for programmable probing function: Q115 to Q118

Parameters Q115 to Q118 contain the **uncompensated** position measurements (i.e. length and radius of the electrode are neglected) that were acquired with the programmable probing function "surface = datum":

Measurement:	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IV axis	Q118

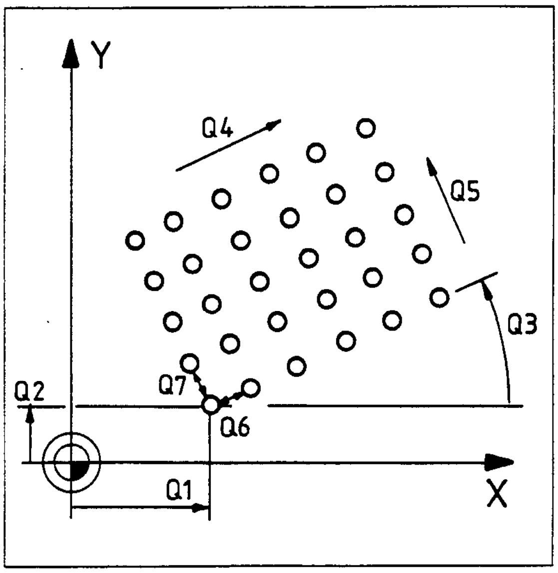
Parametric Programming
Example: Hole pattern

Task

A rotated hole pattern is to be drilled on a surface.

Example:

Starting point:	Q10 = 40 mm	Q2 = 10 mm
Rotational angle:	Q3 = 20°	
Number of holes:	Q4 = 6	
	Q5 = 5	
Distance X axis:	Q6 = 8	
Distance Y axis:	Q7 = 10	



Program	BEGIN PGM 100 MM	
	:	Generator definition
	:	Tool definition, tool call
	FN 0: Q1 = +40	X coordinate of the starting point
	FN 0: Q2 = +10	Y coordinate of the starting point
	FN 0: Q3 = +20	Rotational angle
	FN 0: Q4 = +6	Number of holes in X
	FN 0: Q5 = +5	Number of holes in Y
	FN 0: Q6 = +8	Distance between holes in X
	FN 0: Q7 = +10	Distance between holes in Y
	FN 3: Q9 = +Q4 * +Q6	Distance between first and last hole on the first axis
	CYCL DEF 7.0 DATUM SHIFT	
	CYCL DEF 7.1 X+Q1	Shift datum to the start point
	CYCL DEF 7.2 Y+Q2	
	CYCL DEF 10.0 ROTATION	
	CYCL DEF 10.1 ROT +Q3	Rotation of the coordinate system by angle Q3
	L X+0 Y+0 Z+2 R0	Traverse to start point
	LBL 1	
	FN 0: Q10 = +Q4	Auxiliary parameter (number of holes in the first axis)
	CALL LBL 2	Subprogram call for holes in the first axis
	L IY+Q7	Traverse to second row
	FN 2: Q5 = Q5 − +1	Count down the lines
	FN 11: IF +Q5 GT +0 GOTO LBL	Conditional jump to LBL 1
	CYCL DEF 10.0 ROTATION	Cancel rotation
	CYCL DEF 10.1 ROT+0	
	CYCL DEF 7.0 DATUM SHIFT	
	CYCL DEF 7.1 X+0	Cancel datum shift
	CYCL DEF 7.2 Y+0	
	Z+100 FMAX M02	Retract
Subprogram	LBL 2	
	Z−10 M36	Eroding a hole
	Z+2 FMAX M37	Retract
	L IX+Q6	Traverse to next hole
	FN 2: Q10 = +Q10 − +1	Count down number of holes
	FN 11: IF +Q10 GT +0 GOTO LBL 2	Conditional jump
	L IX − Q9	Retract to starting position
	LBL 0	End subprogram
	END PGM 100 MM	

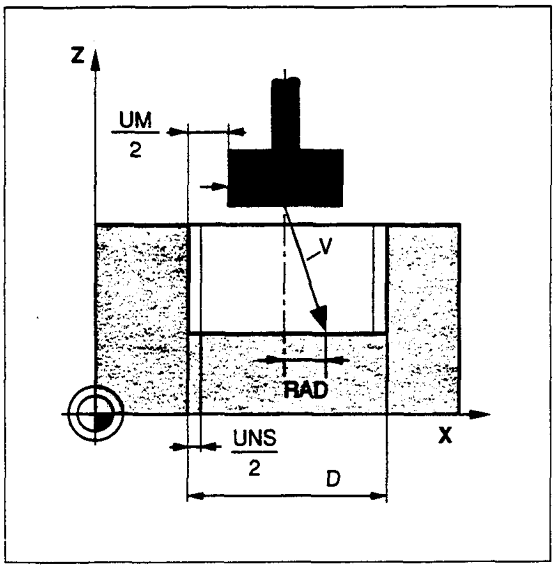
Parametric Programming
Example: Disk

Task

Calculation of an undersize UM and the expansion radius RAD. Erosion of a cavity using the disk cycle (MOD 0).

Example:
Actual electrode radius
Re = R from TOOL DEF: Q108 = 12 mm
Power stage NR: Q99 = 10
Minimum undersize UNS
(from the erosion table): Q231 to Q255
Depth T of the cavity: Q1 = –10 mm
Diameter D of the cavity: Q2 = 40 mm

Starting point
X coordinate = 50
Y coordinate = 50
Z coordinate = 2



Formulas

For the undersize UM:
$UM = D - 2\,RE$

For the expansion radius RAD:
$RAD = \frac{UM - UNS}{2}$

Since the minimum undersize UNS depends on the respective power stage NR, the control must address UNS from NR using indexed data assignment.

UNS = Q230 (Q99)

Since the power stages are numbered 1 to 25, the base parameter Q231 must be reduced by the decrement 1, therefore Q230.

The depth T of the cavity is programmed in the disk cycle, reduced by the minimum undersize UNS.

Note

When choosing the electrode, ensure that the electrode radius Re is greater than the radius $\frac{D}{2}$ of the cavity.

Parametric Programming
Example: Disk

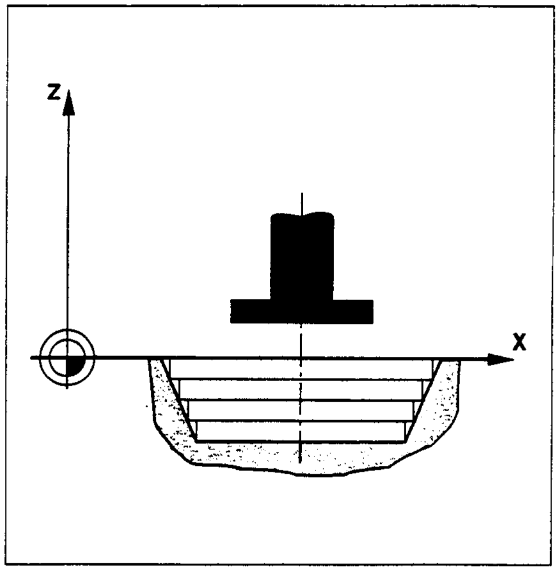
Program

Program	
0 BEGIN PGM 101 MM	
1 CYCL DEF 1.0 GENERATOR	Generator definition
2 CYCL DEF 1.1 P-TAB 300	
3 CYCL DEF 1.2 MAX = 25 MIN = 1	
4 TOOL DEF 1 L+0 R+12	Tool definition: R = Re
5 TOOL CALL 1 Z UM+0	Tool call (activation of Q108)
6 FN0: Q1 = -10	Cavity depth T
7 FN0: Q2 = +40	Cavity diameter D
8 FN0: Q99 = 20	Select power stage NR
9 L X+50 Y+50 Z+2 R0 FMAX	Move to starting point
10 FN3: Q10 = +2 * + Q108	Calculate undersize UM
11 FN2: Q10 = +Q2 - +Q10	
12 TOOL CALL1 Z UM +Q10	Tool call
13 FN16: Q11 = Q230 (Q99)	Find present minimum undersize
14 FN2: Q12 = +Q10 - +Q11	Calculate expansion radius RAD
15 FN4: Q12 = +Q12 DIV +2	
16 FN1: Q13 = +Q1 + +Q11	Calculate depth T-UNS
17 CYCL DEF 17.0 DISK	
18 CYCL DEF 17.1 Z + Q13	Sink to depth T-UNS
M36	Generator on
19 CYCL DEF 17.2 RAD = Q12	Incremental increase of expansion radius Q12
MOD = 0	
20 L Z+ 100 R0 FMAX M37	Retract electrode, Generator off
21 END PGM 101 MM	

Parametric-Programming
Example: Conical cavity

Task

Conical cavity, machined using program 101 (disk).



Changes in PGM 101

To use program 101 (disk), lines 1 to 9 and 20 must first be erased.

Example:
Actual electrode radius
Re = R from TOOL DEF: Q108 = 12 mm
Power stage NR: Q99 = 20

Depth T of the conical cavity: Q0 = −20
Infeed ΔT (depth per disk): Q1 = −1
Upper diameter D1: Q2 = 40
Lower diameter D2: Q3 = 30

Starting point
X-coordinate = 50
Y-coordinate = 50
Z-coordinate = 2

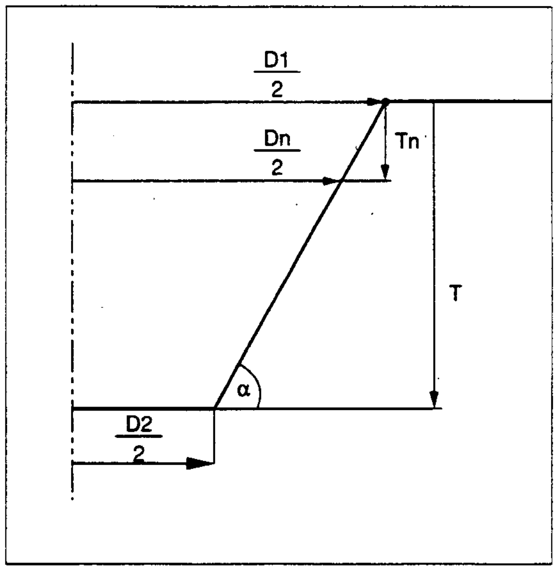
Formulas

The geometrical context of the depth T and the two diameters provide the angle α at the wall of the conical cavity:

$$\alpha = \arctan \frac{T}{\frac{D1 - D2}{2}}$$

The respective diameter Dn at a certain depth Tn is calculated as follows:

$Dn = D1 - \cos\alpha \cdot Tn$



Note

Be sure to choose an electrode appropriate to the lower diameter D2 and upper radius $\frac{D1}{2}$.

Parametric Programming
Example: Conical cavity

Program

Program	
0 BEGIN PGM 102 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y-100 Z+0	
3 TOOL DEF 1 L+0 R+12	Tool definition: R = Re
4 TOOL CALL 1 Z U+0	Tool call (activation of Q108)
5 CYCL DEF 1.0 GENERATOR	Generator definition
6 CYCL DEF 1.1 P-TAB 300	
7 CYCL DEF 1.2 MAX = 25 MIN = 1	
8 FN0: Q0 = -20	Depth T of the conical cavity
9 FN0: Q1 = -1	Infeed ΔT
10 FN0: Q2 = +40	Upper diameter D1
11 FN0: Q3 = +30	Lower diameter D2
12 FN0: Q99 = +10	Select power stage NR
13 L X+50 Y+50 Z+1 R0 FMAX	Move to starting point
14 FN0: Q20 = + Q0	Store Q parameters Q0 to Q2 for further calculation!
15 FN0: Q21 = +Q1	
16 FN0: Q22 = +Q2	
17 FN8: Q20 = Q20 LEN + 0	Form value of depth T
18 FN2: Q30 = +Q22 - +Q3	Find cos α for further calculations
19 FN4: Q30 = +Q30 DIV +2	
20 FN13: Q30 = +Q20 ANG +Q30	
21 FN7: Q30 = cos + Q30	
22 FN0: Q32 = +0	Start value for positioning
23 FN0: Q33 = +0	and set depth Tn (also abort criterion)
24 LBL 1	Loop beginning
25 FN1: Q23 = +Q33 + +21	Calculate respective depth Tn
26 FN12: IF +Q33 LT +Q0 GOTO LBL 2	Depth Tn already smaller than final depth T?
27 FN9: IF +0 EQU +0 GOTO LBL 3	If no, jump to marker 3
28 LBL 2	If Tn is smaller than T
29 FN0: Q33 = +Q0	set Tn = T
30 LBL 3	
31 FN3: Q34 = +Q30 * -Q32	For respective depth Tn calculate
32 FN2: Q34 = +Q22 - +Q34	corresponding diameter Dn
33 FN0: Q1 = +Q33	Load transfer parameter for program 101
34 FN0: Q2 = + Q34	with Tn and Dn
35 CALL PGM 101	Call program 101
36 FN0: Q32 = +Q13	Position to erosion depth
37 L Z+ Q31 R0	
38 FN9: IF +Q33 EQU +Q0 GOTO LBL 99	If depth T is reached, end loop
39 FN11: IF +Q33 GT +Q0 GOTO LBL 1	If depth T is not reached, continue loop
40 LBL 99	
41 LZ +100 R0 FMAX M37	Retract, Generator off
42 END PGM 102	

Programmed Probing
Overview

The programmable probing function allows you to perform measurements before or during machining. For example, the surfaces of parts with different heights can be probed, so that the correct depth is always attained during subsequent machining.

In addition, thermally-induced position deviations of the machine can be determined at selected time intervals and compensated.

Process

First, pilot position in rapid traverse while maintaining the setup clearance (machine parameter). Then probe with the probing axis at the measuring feed rate, transfer the probing position and retract to the setup clearance in rapid traverse. If the short circuit signal is not released before reaching the maximum probing depth (machine parameter), the probing operation is aborted.

Initiate the dialog TOUCH PROBE

Input

PARAMETER NUMBER FOR RESULT ?	ENT Parameter number
PROBING AXIS/PROBING DIRECTION ?	X +/– ENT Probing axis and probing direction
POSITION VALUE ?	X All coordinates of the pilot position Y ENT or incremental

Example

The electrode moves to starting position X–10, Y+20 and Z–20, and then begins probing with the X axis in positive direction. The probed result (X position) is stored in Q10.

Program

TOOL DEF 1 L+R R+5	
TOOL CALL 1 Z U+0	
L Z+200 R0 FMAX M06	Tool change position
TCH PROBE 0.0 REF.PLANE Q10 X+	Probing with the X axis in positive direction
TCH PROBE 0.1 X–10 Y+20 Z–20	Move to starting position Q10 contains the compensated X axis measurement after probing

Q115 to Q118

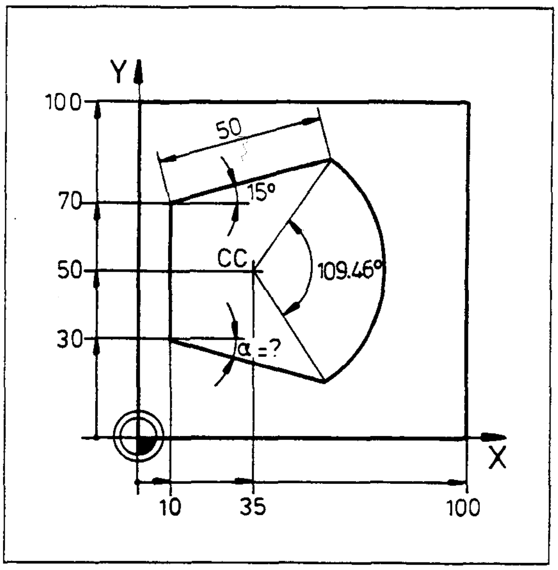
After the probing Q115 to Q118 contain the actual, uncompensated values for X, Y, Z and IV.

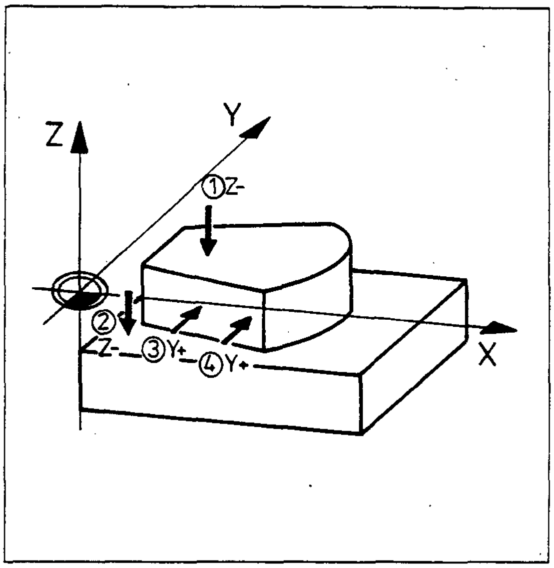
TOUCH PROBE Q

Programmed Probing
Example: Measuring length and angle

Task

A height (from the probing points ① and ②) and an angle (from the probing points ③ and ④) are to be measured with parameter programming.





Main program: Definition of probing points (pilot positioning)

0 BEGIN PGM PROBE MM

1 FN 0: Q11 = +20	Probing point ①
2 FN 0: Q12 = +50	X, Y, Z coordinates for
3 FN 0: Q13 = +10	pilot positioning
4 FN 0: Q21 = +20	Probing point ②
5 FN 0: Q22 = +15	
6 FN 0: Q23 = +0	
7 FN 0: Q31 = +20	Probing point ③
8 FN 0: Q32 = +15	Z coordinate Q33 valid
9 FN 0: Q33 = −10	for probing point ④
10 FN 0: Q41 = +50	Probing point ④
11 FN 0: Q42 = +10	
12 TOOL DEF 1 L+0 R+5	
13 TOOL CALL 0 Z	
14 L Z+100 R0 F1000 M6	Retract, insert electrode

Measure length

15 TCH PROBE 0.0 REF.PLANE Q10 Z–	① Probe
16 TCH PROBE 0.1 X+Q11 Y+Q12 Z+Q13	
17 L Y+Q22	Approach auxiliary point
18 TCH PROBE 0.0 PROBE Q20 Z–	② Probe
19 TCH PROBE 0.1 X+Q21 Y+Q22 Z+Q23	
20 CALL LBL 1	Call subprogram 1

Measure angle

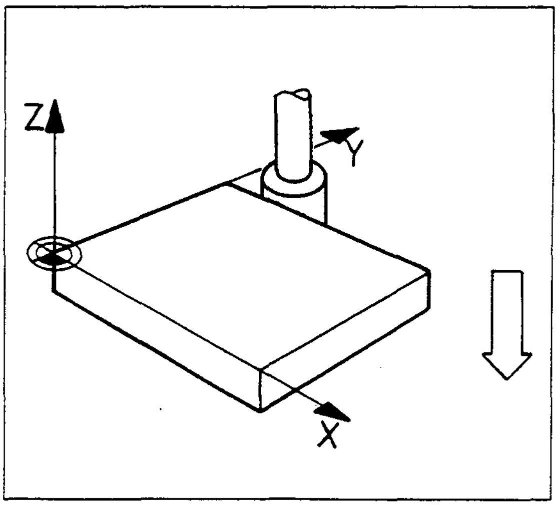
21 TCH PROBE 0.0 REF.PLANE Q30 Y+	③ Probe
22 TCH PROBE 0.1 X+Q31 Y+Q32 Z+Q33	
23 TCH PROBE 0.0 REF.PLANE Q40 Y–	④ Probe
24 TCH PROBE 0.1 X+Q41 Y+Q42 Z+Q33	
25 CALL LBL 2	Call subprogram 2
26 STOP	Check result parameter (see chapter "Machine Operating Modes", section "Program run", Checking/Changing the Q Parameters)
27 L Z+100 R0 F1000 M2	Retract, jump to start of program

Programmed Probing
Example: Measuring length and angle

Subprogram 1: measure length	28 LBL 1 29 FN 2: Q1 = +Q20 – +Q10 30 LBL 0	 Measured **height** in parameter **Q1.**
Subprogram 2: measure angle	31 LBL 2 32 FN 2: Q34 = +Q40 – +Q30 33 FN 2: Q35 = +Q41 – +Q31 34 FN 13: Q2 = +Q34 ANG+Q35 35 FN 1: Q2 = –360 + +Q2 36 LBL 0 37 END PGM PROBE MM	 Measured **angle** in parameter **Q2.**

Transfer actual position

The actual tool position can be transferred to the machining program with the "Transfer actual position" key.



Application possibilities

or

X R+ Z

or

TOOL DEF

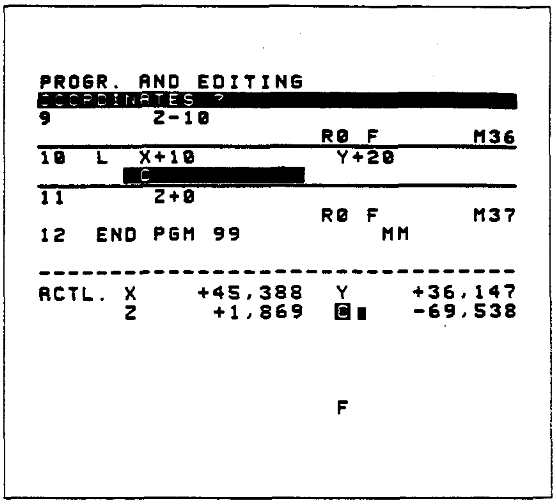
In this way you can transfer:
- positions
- tool dimensions (see Tool Definition).

Process

Traverse the tool to the desired position.

Open a program block (e.g. for a straight line) in the "Programming and editing" operating mode. Select the axis from which the actual value is to be transferred.

This axis position is transferred by pressing the "Transfer actual position" key.



Example

X Y Z Move the axis or axes via the axis keys.

Input

Initiate the dialog

COORDINATES ?	X Y ...	Transfer the axis positions individually.
	ENT	Terminate position input.

RADIUS COMP.: RL/RR/NO COMP. ?	RL	Enter the radius compensation if needed.
FEED RATE ? F=	ENT	Enter the feed rate, if needed, and confirm entry.
MISCELLANEOUS FUNCTIONS M ?	ENT	Enter miscellaneous function if needed.

You can skip dialog queries with "NO ENT" or "END □".

In the "Test run" operating mode, a machining program is checked for the following errors without machine movement:

- Overrunning the traversing range of the machine
- Illogical entries, e.g. redundant input of one axis
- Failure to comply with elementary programming rules e.g. cycle call without a cycle definition
- Certain geometrical incompatibilities

The lowest line on the screen shows the tool number (T1), the tool axis (Z), the undersize (UM), the feed rate (F) and the M functions.

The line above it shows the distance remaining to be eroded (WTG for "Way To Go").

```
TEST RUN                          100
       TO BLOCK NUMBER =
11   FN 0:      Q1 = +40
12   FN 0:      Q2 = +10
13   FN 0:      Q3 = +20
14   FN 0:      Q4 = +6
----------- 00:00:00 -----------
NOML. X    -10,000   Y      -10,000
      Z    +10,000   C       +0,000
300/25-1/25          WTG     0,000

T9999 Z UM 0,100     F   1000   M37
```

Testing the program

Initiate the dialog — PGM NR

PROGAM SELECTION PROGRAM NUMBER =	ENT — Select the program to be tested.
TO BLOCK NUMBER =	ENT — Key in and transfer the block number up to which the test is to run. or NO ENT — Test the complete program.

No apparent errors

If the program contains no apparent errors, the program test runs until the entered block number is reached, or a jump is made back to the start of program if no STOP or M06 was programmed.

STOP/M06

If a STOP or M06 was programmed, the test can be continued by entering a new block number or by pressing the "NO ENT" key.

Error

If an error is found, the program test is stopped. The error is usually located in or before the stopped block. An error message is displayed on the screen.

STOP — The program test can be halted with the "STOP" key and aborted at any time.

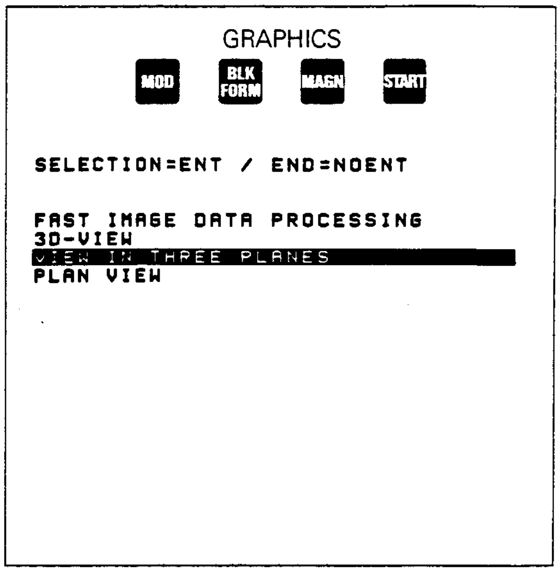
The program is tested block by block.

MOD — With the MOD key you can set the position display to show nominal values (NOML) to enable you to check the programmed position coordinates.

Test Graphics

Machining programs can be simulated graphically and tested if a blank has been previously defined (BLK FORM).

More information on the definition of the blank can be found in the section Program Selection, Blank form definition.



MOD GRAPHICS

After selecting a program, the "menu" shown at the right is displayed by pressing the GRAPHICS "MOD" key twice.

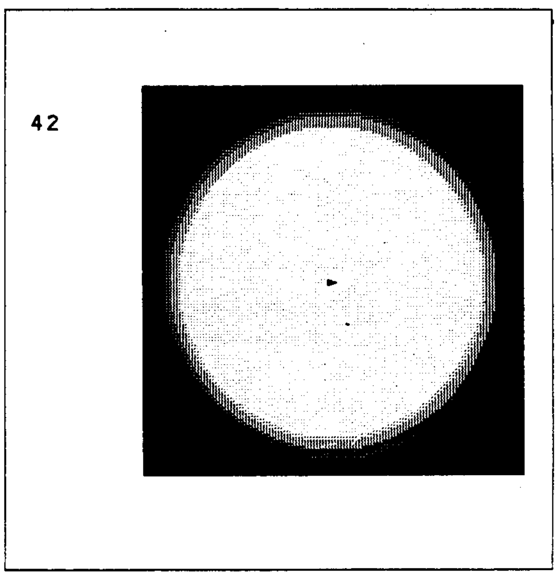
One of the versions of the graphic presentations can be selected with the vertical cursor keys and entered with the "ENT" key.

START

The graphic simulation or internal computation is started with the "START" key.

Fast data image processing

With **"Fast data image processing"** only the current block number is displayed on the screen and the internal computing also indicated by an asterisk (* = control is started).
When the program has been processed, the "machined" workpiece can be displayed in plan view, view in three planes or 3D view.

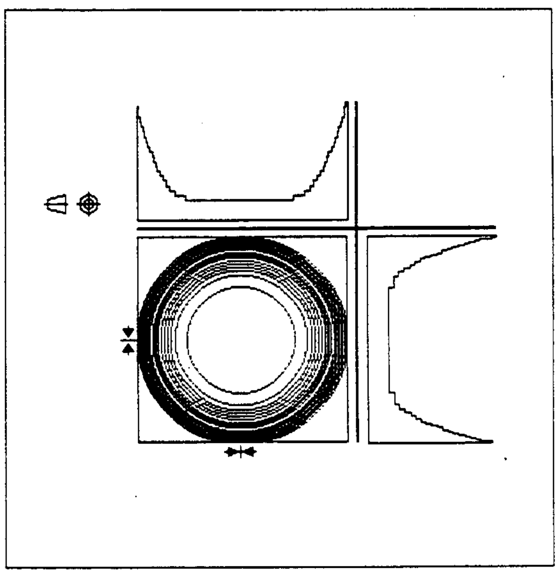


Plan view with depth indication

The workpiece center is shown in the **plan view** with up to 7 different shades: the lower the contour, the darker the shade.

View in three planes

The **view in three planes** shows the workpiece – as in drafting – with a plan view and two sections.



The sectional planes can be moved via the cursor keys.

The view in three planes can be switched from the German to the English preferred projection via a machine parameter. A symbol (in conformance to ISO 6433) indicates the type of projection:

European preferred:

American preferred:

Test Graphics

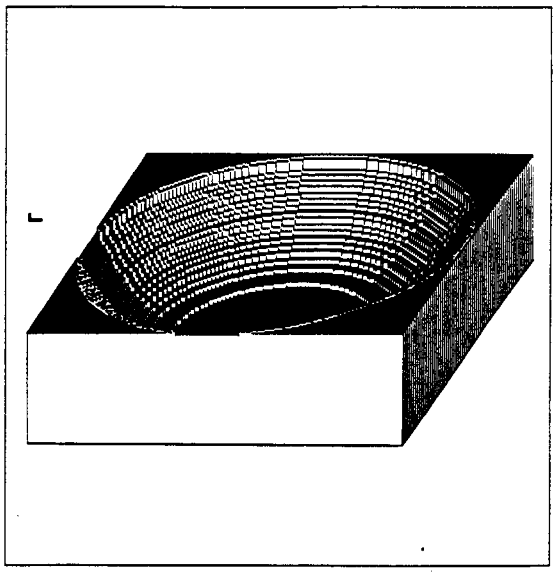
3D view

The program is simulated in a three-dimensional view.

The displayed workpiece can be rotated by 90° with each activation of the horizontal cursor keys. The orientation is indicated by an angle.

└ = 0°   ┐ = 180°
┘ = 90°   ┌ = 270°

If the height to side proportion is between 0.5 and 50, the type of display can be changed with the vertical cursor keys. You can switch between a scaled and non-scaled view. The short height or side is shown with a better resolution in the nonscaled view. The dimensions of the angle indicator change to a show the disproportion.



Magnifying

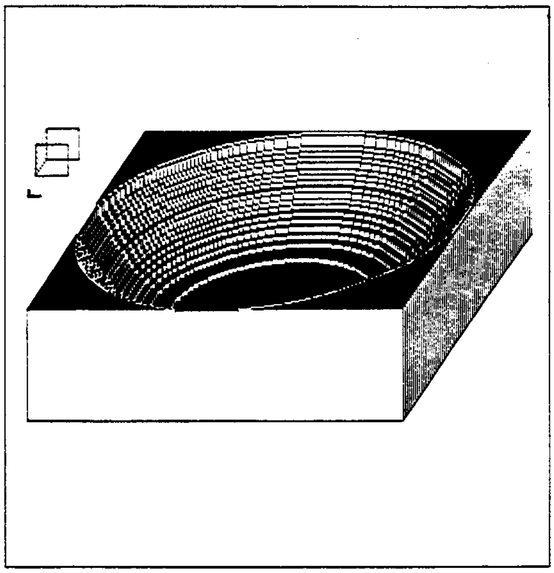
You can magnify a detail of the displayed work with the "MAGN" key. A wire model with a hatched surface appears next to the graphic. This marks the sectional plane.

Selecting the sectional plane

You can select a different sectional plane with the horizontal cursor keys.

Trimming

You can trim the selected plane or cancel the section with the horizontal cursor keys.



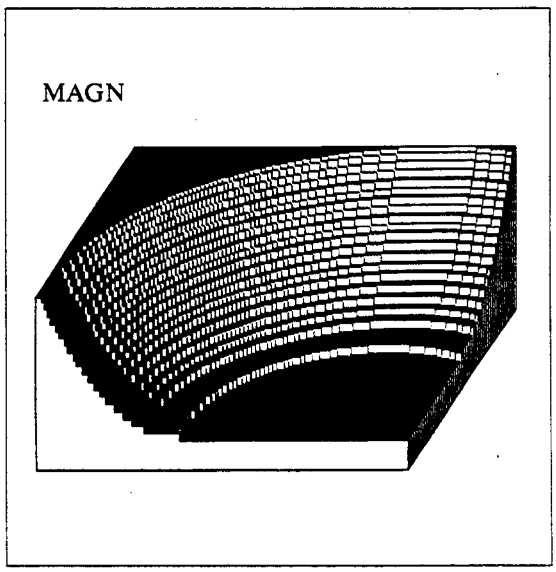
Transferring the detail

Once the desired detail is displayed, select the dialog "TRANSFER DETAIL = ENT" with the vertical cursor keys and confirm with the "ENT" key.

Magnification

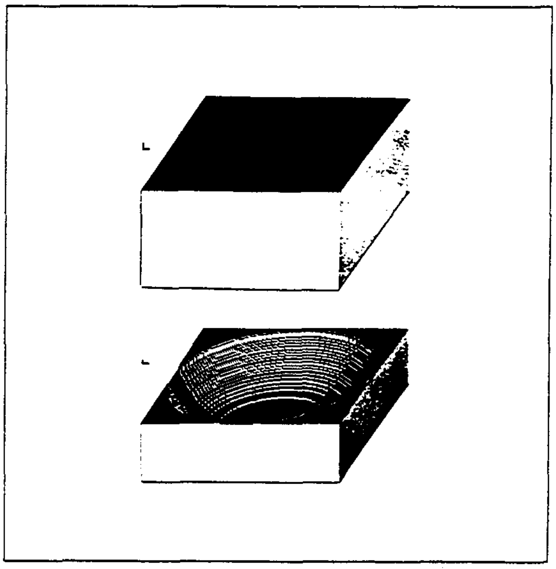
The "remaining workpiece" is displayed on the screen with "MAGN".

Another graphic simulation of the magnified detail can be executed in the plan view, the view in three planes or the 3D view via the "START" key.



Test-Graphics

You can revert to the blank with the "BLK FORM" key and restart simulation with "START".



Tips

The "3D view" and "View in three planes" require extensive computing. For long programs, we therefore recommend displaying the workpiece with "Fast data image processing" or in the quicker "Plan view with depth indication" first, and then switching to the "3D view" or the "View in three planes".

Displaying details

The following aids are available if fine details are to be examined:

- Trim the blank and magnify in an additional graphic program run.
- Restrict the blank detail to the section of interest.

Tool call

One "TOOL CALL" must be programmed prior to the first axis movement to designate the tool axis. Specifying the tool axis in the BLK FORM definition does not suffice for the graphic program run. Both entries for the axis must be the same.

If the tool axis is not given, an error message appears after starting the graphics.

Counting the Machining Time

In the program run/full sequence mode the machining time is shown above the status display.

It is also possible to count time in a timer table with the name TIME.W. In this table the programmed power stages and the corresponding machining times per power stage appear in the first and second columns, respectively. A third column shows the absolute times at which each new power stage begins.

Open the table TIME.W with the code number 963 in the programming and editing mode. Then use the CYCL DEF key to insert as many lines as there are power stages in the part program.

When a new program is started the old values in the table are overwritten.

```
ERODING TIME TABLE TIME
POWER STAGE NUMBER
    TAB:TIME                    NR:10
NR  REL.        ABS.
10  00:01:10    00:00:00
8   00:01:19    00:01:10
6   00:00:52    00:02:29
4   00:00:57    00:03:22
0   00:00:00    00:04:19
0   00:00:00    00:00:00
-----------------------------------------
ACTL. X      +0,013   Y         +0,018
      Z      +0,018   C         +0,015

                          F  0
```

External Data Transfer
Transfer menu

Part programs can be read into or out of the control in the "Programming and editing" mode of operation.

Data transfer must be started from the control. Files are best transferred in a blockwise manner using the FE 1 transfer mode and the FE 401B floppy disk unit or the HEIDENHAIN data transfer software.

Selecting the transfer menu

The menu items in the transfer menu are selected by pressing the cursor keys and pressing ENT.

```
PROGRAMMING AND EDITING
SELECTION = ENT

READ-OUT ALL PROGRAMS
READ-IN ALL PROGRAMS
READ-OUT SELECTED PROGRAM
READ-IN SELECTED PROGRAM
CLEAR SELECTED PROGRAM
FURTHER FUNCTIONS
```

Exiting the transfer menu and functions

All transfer menus can be exited at any time by pressing END □.

Read-in/ Read out

The first four menu items are for importing and exporting files.

Selecting the read-in function calls an external file directory to screen (EXT-FILES:); the read-out function calls the internal file directory (INT-FILES).

```
READ-OUT SELECTED PROGRAM
OUTPUT = ENT

0   . D  432
10  . H  774 * R
505 . H  216
80  . H  72      E
910 . E  900 * R
911 . E  324
```

Interrupt data transfer

Data transfer can be interrupted at any time by pressing END □.

An interruption releases the error message:
PROGRAM INCOMPLETE

Erasing files

Selecting the "Clear selected program" function calls the external file directory to screen.

Use the cursor keys to select the unwanted file, then press ENT.

```
C:\PGM\TNC 306
CLEAR = ENT

0   . D  2
10  . H  3
505 . H  1
80  . H  1
910 . E  6
911 . E  1
```

Selecting further functions

Selecting the menu item "Further functions" calls the additional menu illustrated at right. The items are selected as in the main menu, i.e. by moving the highlight with the cursor keys and pressing ENT.

```
FURTHER FUNCTIONS
OUTPUT = ENT

PRINT FILE
FILE PROTECTION
FORMAT DISK
```

Printing a file

If a printer is connected to PRINT interface of the FE 401 or to the parallel interface of a PC with the HEIDENHAIN data transfer software, the file selected in the external file directory can be printed.

File protection

Files in the external storage medium can be protected from unintentional erasure.
This file protection can be canceled by pressing NO ENT.

```
FILE PROTECTION
YES = ENT/NO = NO ENT

0   . D  P 1
10  . H    3
505 . H    2
80  . H    1
910 . E  P 6
911 . E  P 1
```

Formatting floppy disks

Floppy disks located in the upper disk drive of the FE 401 floppy disk unit can also be formatted from the TNC.

External Data Transfer
FE 401B Floppy Disk Unit
Data transfer software from HEIDENHAIN

Preparation

Both the FE 401B floppy disk unit and the HEIDENHAIN data transfer software must be prepared according to the operating instructions.

Both the FE 401B floppy disk unit and the HEIDENHAIN data transfer software are adjusted to the TNC 306 and are therefore very easy to put into service.

FE 401B floppy disk unit

- Connect power cable of the FE 401B.
- Connect data transfer cable.
- Set FE 401B power switch to ON position.
- Insert floppy disk (3.5" DS, DD 1.0M byte) into upper disk drive. The floppy disk must be formatted and must not be write protected.
- If necessary, select baud rate and V.24 interface mode (FE or ME).

HEIDENHAIN data transfer software

- Installation on PC after purchasing the software.
- Prepare PC.
- Connect data transfer cable.
- Switch on PC and start data transfer software.
- If necessary, select the baud rate and file path.

TNC 306

- If necessary, select baud rate and RS-232-C interface mode FE 1 via MOD functions.

Example

Read-in selected program

Selection

READ-IN SELECTED PROGRAM	Select function and confirm selection.

In the RS-232-C interface mode FE 1 the external file directory is read-in and displayed.

READ-IN = ENT	Select desired file and read-in. Either read in more files or terminate data transfer.

External Data Transfer
Non-HEIDENHAIN devices/software

Preparation

If you wish to use non-HEIDENHAIN devices or software, the RS-232-C interface of the TNC must be adjusted to the EXT mode (see section "MOD Functions", chapter "General Information").

Machine parameters 5010 to 5020 are to be adjusted to the non-HEIDENHAIN device or software.

EXT

If the RS-232-C mode **EXT** is selected, the TNC is operating with a **standard data interface.**

Adjusting non-HEIDENHAIN devices/software

The interface descriptions of all units and programs must be matched.

We recommend the following procedure:

- Find the common settings (e.g. data format, baud rate).
- Wire the data transfer cable to its connector according to the proper pin layout.
- Connect the data transfer cable.
- Connect the power cable of the peripheral device.
- Switch on power.
- If necessary, start the data transfer program on the PC.

Note

The following description of machine parameters MP 5010 to MP 5020 is only applicable when the data interface is in the EXT mode. See chapter "General Information", section MOD Functions, User parameters for instructions for calling user parameters.

MP 5010
End of file character

The machine parameter MP 5010 determines the control character for End of Text (ETX) for input or output (e.g. MP 5010 = 3: ETX).
If MP 5010 = 0, no end of file control character will be transmitted!

MP 5011
Interrupt character

Machine parameter MP 5011 defines the control character for End of Transmission (EOT) (e.g. MP 5011 = 4: EOT).
If MP 5011 = 0, no interrupt control character will be transmitted!

MP 5020 defines the data format and the type of transmission stop.

MP 5020
Data format

Function	**Bit**	**Input**	**Input values**
7 or 8 data bits	0	+ 0 → 7 data bits (ASCII-code with 8^{th} bit = parity) + 1 → 8 data bits (ASCII-Code with 8^{th} bit = 0 and 9^{th} bit = parity)	1
No function	1	+ 0	0
Transfer stop due to RTS	2	+ 0 → inactive + 4 → active	0
Transfer stop due to DC3	3	+ 0 → inactive + 8 → active	8
Character parity even or odd	4	+ 0 → even + 16 → odd	0
Character parity required	5	+ 0 → not required + 32 → required	32
No function	6	+ 0	0
Number of stop bits	7	+ 0 → 2 stop bits + 128 → 1 stop bit	128
		Value to be entered for MP 5020:	169

Example of value determination

Standard data format: 7 data bits (ASCII code with 7 bits, even parity) Transfer stop due to DC3, 1 stop bit

Bits 0 – 7	**7**	**6**	**5**	**4**	**3**	**2**	**1**	**0**
Significance of bit	128	64	32	16	8	4	2	1
Enter 0 or 1 accordingly	1	0	1	0	1	0	0	0

After adding the significances, you obtain the input value for machine parameter 5020.
In our example: 168.

MP 5990
Block number

Block number check for external data transfer:
0 ≙ Check block numbers
1 ≙ Do not check

Miscellaneous functions M

Miscellaneous functions with predetermined function

	Function	Effective at Begin of block	Effective at End of block	Reference Page
M00	Stop program run		●	P22
M02	Stop program run/or clear the status display/return jump to block 1		●	P22
M03	Non-controlled rotation of C axis[1]	●		P21
M04	Non-controlled rotation of C axis[1]	●		P21
M05	Stop non-controlled rotation of C axis		●	P21
M06	Electrode change/or stop program run/stop spindle		●	P22
M08	Flushing on	●		P21
M09	Flushing off		●	P21
M30	like M02		●	P22
M36	Erosion ON (gap control on)	●		P21
M37	Erosion OFF (gap control off)	●		P21
M38	Transferring coordinate values from the datum table 0.D to a part program	●		P84
M39	Transferring Q parameter values from a part program to the datum table 0.D	●		P84
M89	Vacant miscellaneous function	●		
M89	Cycle call, modal (depends on machine parameters)		●	P64
M91	**in the positioning block:** Coordinates refer to the scale reference point	●		P56
M92	**in the positioning block:** Coordinates refer to a position defined by the machine manufacturer (machine datum), e.g. tool change position	●		P56
M93	Reserved	●		
M95	Reserved		●	
M96	Reserved		●	
M97	Path offset on outside corners: Intersection instead of tangential circle		●	P54
M98	Blockwise end of path offset		●	P55
M99	Cycle call effective blockwise		●	P64

[1] The direction of rotation is determined by the machine tool builder.

Program example: EDM machining

Dialog initiation key	Function	Example values
PGM NR	Program number, mm/inch	BEGIN PGM 729 MM
BLK FORM	Blank definition: tool axis Minimum point Maximum point	BLK FORM 0.1 ZX+0 Y+0 Z–40 BLK FORM 0.2 X+100 Y+100 Z+0
CYCL DEF	Calling the desired erosion parameter table	CYCL DEF 1.0 GENERATOR CYCL DEF 1.1 P-TAB 10 CYCL DEF 1.2 MAX = 8 MIN = 1
TOOL DEF	Electrode definition Electrode number Electrode length Electrode radius	 TOOL DEF 1 L+0 R 7.5
TOOL CALL	Electrode call Electrode number Tool axis (e.g. Z), undersize (UM)	 TOOL CALL 1 Z UM
	Electrode change Retract tool axis, length compensated, Radius uncompensated, electrode change	 L Z+200 R0 M06
	Starting position: Approach the workpiece No radius compensation	L X–20 Y–20 R0
	Move tool axis to working depth	L Z–20
	Machine the work, approach 1st contour point with radius compensation **Generator ON** : **MACHINING** :	L X+0 Y+0 RL M36
	Last contour point (with radius compensation)	L X+0 Y+0 RL
	After machining Retract in the machining plane Deselect radius compensation **Generator OFF**	 L X–20 Y–20 R0 F500 M37
	Retract the tool axis, Return jump to the 1st block	L Z+200 M02

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
D-83301 Traunreut, Deutschland
☏ (0 86 69) 31-0 · Tx 56 831
FAX (0 86 69) 50 61

281 388 20 · SW04 · 1 · 6/97 · S · Printed in Germany · Subject to change without notice